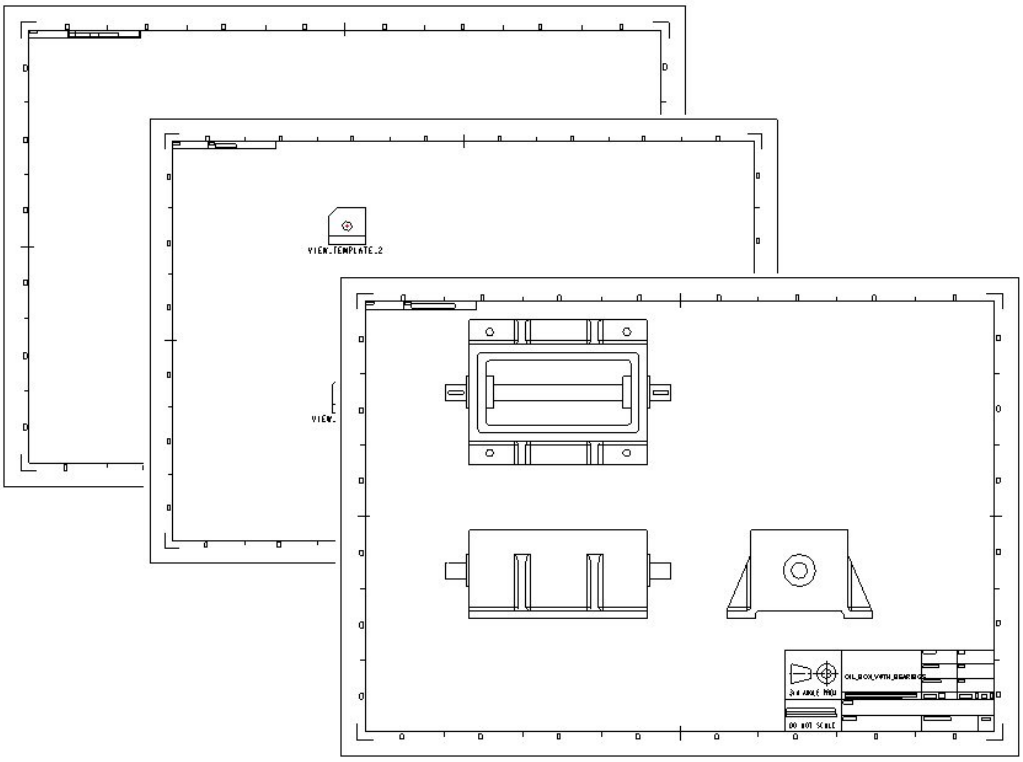
Drawing formats and templates

Creo[tm] Parametric
Education Advanced Edition



design TECHNOLOGY EDUCATION

CONTINUING PROFESSIONAL DEVELOPMENT
Teaching in Scotland

D&T the design and technology association

department for children, schools and families

PTC

C1-SE-L1-J.E.F-10

Drawing formats and templates

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Amended Dec2008 for Creo^{tm} Parametric

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Manufacturing -

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Feedback

In order to ensure these materials are of the highest quality, users are asked to report errors to the author.

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Suggestions for improvements and other activities would also be very welcome.

C1-SE-L1-J.E.F.-10

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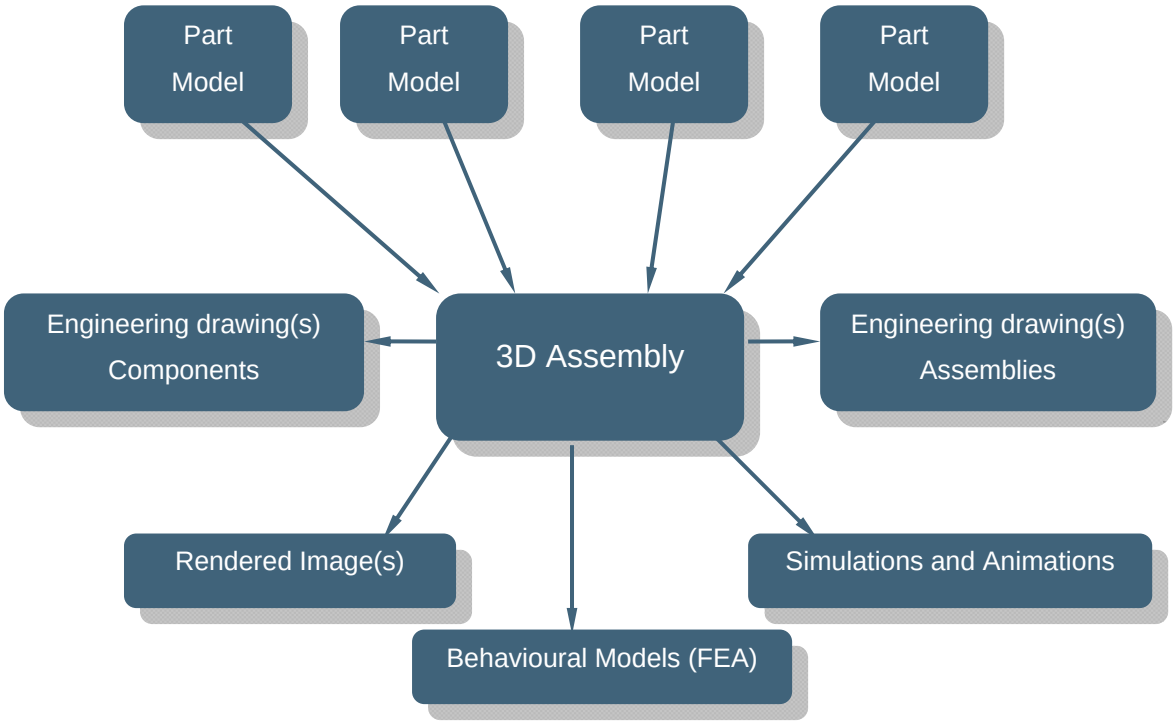
Overview

The creation of an Orthographic set of drawings in Creo[tm] Parametric is very simple once you get the hang of it but can appear daunting when you first approach the procedure.

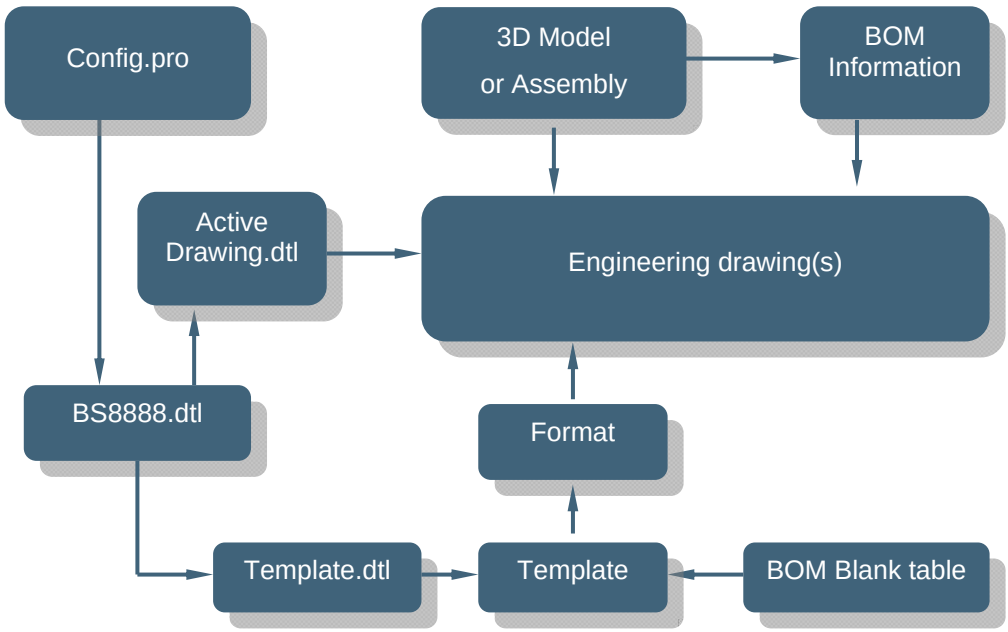
The first thing you have to keep in mind is that what you are creating is a particular way of looking at a three dimensional object.

In the diagram below you can see the relationship between various aspects of the outputs available from Creo[tm] Parametric. The various part models (*.prt) can be assembled to form complex assemblies (*.asm). Both parts and assemblies can be shown as drawings (*.drw). Whilst not part of this tutorial, the models can also be shown as rendered images; simulations and animations; and behavioural models using the Finite Element Analysis (FEA) part of the Schools Advanced Edition.



For Orthographic drawings, often called Detail Drawings, Creo[tm] Parametric merely represents this 3D object using whatever standard you have chosen.

The diagram below summarises the relationship between the 3D model or assembly; the engineering drawings and the various ways that the program creates the drawings and sets the agreed standards for that particular set of orthographic drawings.



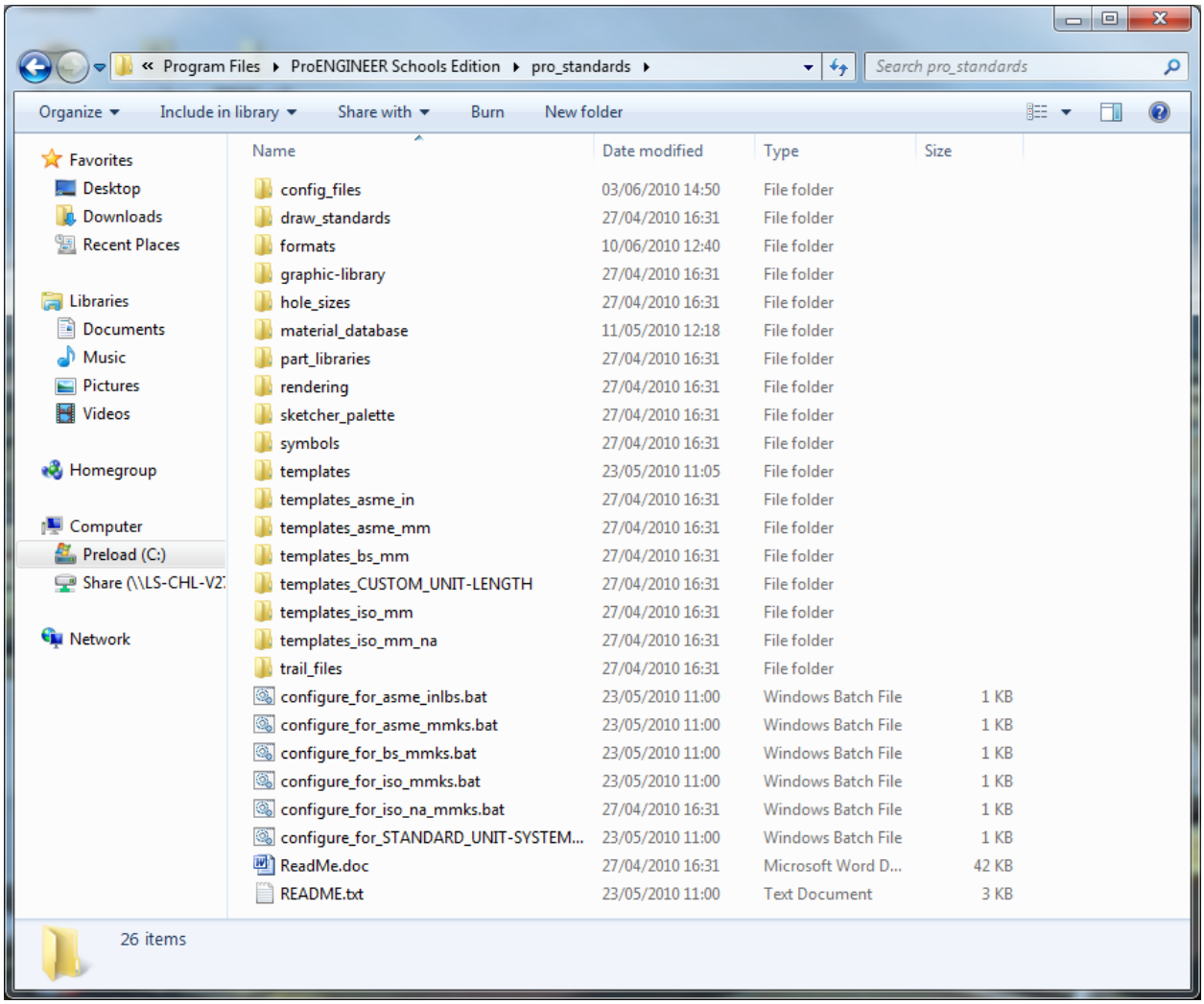
Firstly, make sure you have the correct drawing setup file (*.dtl) referenced in the config.pro file. The standard 'look' of orthographic drawings will vary depending on the part of the world you are in and which particular set of standards you need to use to communicate your ideas with others. In the United Kingdom the British Standards Institute set the agreed standard, the current one for schools being BS8888. In other areas the ISO (International Organization for Standardization) or ANSI (American National Standards Institute) standards apply; if you work in these areas you will need to set the Configuration File to suit your standard.

In Creo^{tm} Parametric there are a number of batch files (*.bat) in the 'pro_standards' folder of the program. Activating the appropriate batch file will set the config.pro file to the correct standard for your needs.

To create a drawing you will also need a Format, this file is the 'look' of your drawings. The Format can also be overlain with a Template that positions the views and creates the text height, line type etc. This is controlled by the *.dtl file which could be the default, in this case the bs8888.dtl file, or a specially created one called the Active Drawing File which will only affect the drawing to which it is attached. If a Bill of Materials (BOM) is required, a 'bom' table can be inserted into the Template that can extract the necessary information from the Bill of Materials contained within the assembly file.

- A drawing format file (*.dtl) can be tailored to a specific set of drawings and saved under a name, e.g. Oil_Box.dtl in which special changes are made to suit the needs of this set of drawing. This is the Active Drawing file.
- If a Template is used its drawing format file (*.dtl) file will determine the outcomes unless overwritten by an Active Drawing file.
- If no template is used then the Global dtl file, e.g. bs8888.dtl, will be used.

To assign a dtl file to the config.pro file



1. Open the creo_standards folder in the C:/Program Files/PTC/ Creo1.0/Common Files/F000. It will look a little like the one above. Read the README.txt file therein (reproduced below)

```
Dear Creo Users,

The PTC Education Group has provided a number of batch files to help
set your default units quickly and easily.

If you choose the "Metric" option during the installation of Creo
Parametric, it is installed using
the MMKS Unit System and ASME drawing standard.

If you choose the "English" option, Creo is installed using the
INLBS Unit System and ASME drawing standard.

You can easily reconfigure your installation to follow a different
Unit System and/or Drawing Standard
using one of the supplied configuration .bat files as follows:

1) Exit Creo Parametric.
```

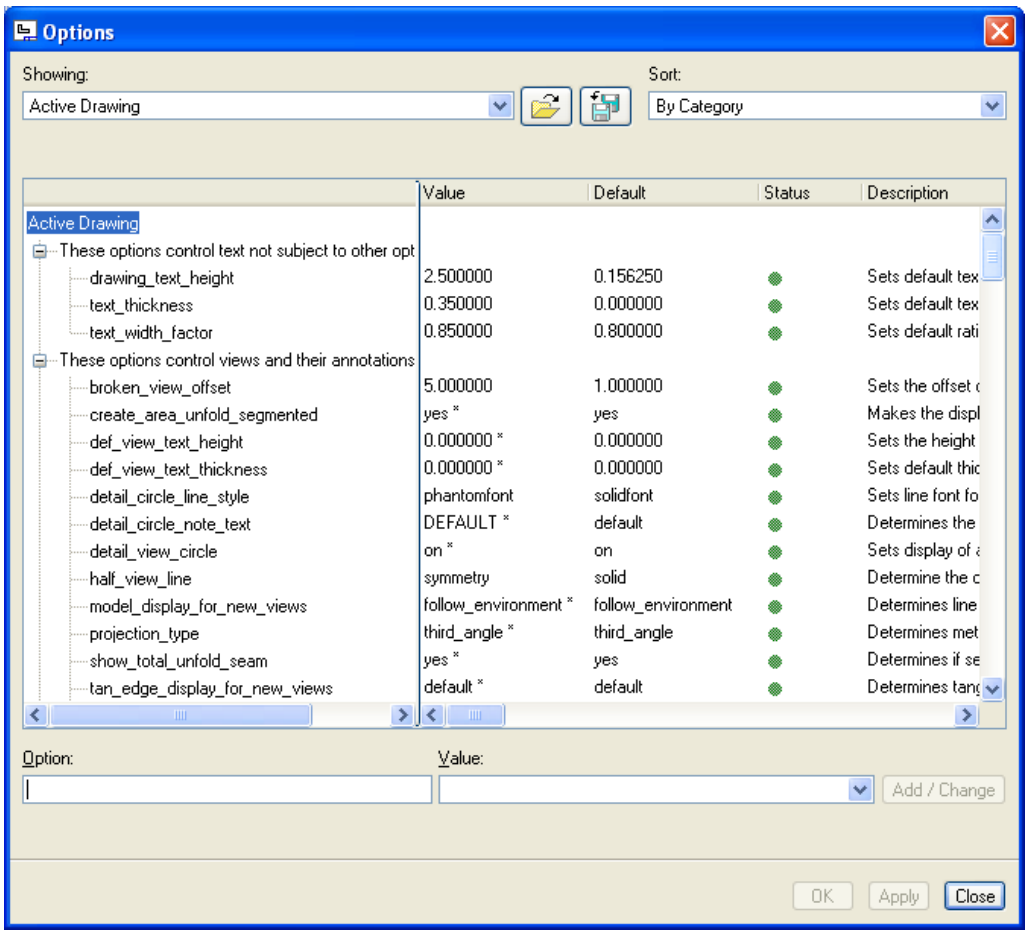
```
2) Double-click one of the five ".bat" files to configure your
Pro/ENGINEER Education installation for a desired
   Unit System and Drawing Standard:

     configure_for_asme_inlbs.bat – ASME standard drawings with INLBS
unit system
     configure_for_asme_mmks.bat – ASME standard drawings with MMKS
unit system
     configure_for_bs_mmks.bat – BS8888 standard drawings and an MMKS
unit system
     configure_for_iso_mmks.bat – ISO standard drawings and an MMKS
unit system (1st Angle Projection & Commas)
     configure_for_STANDARD_UNIT-SYSTEM.bat - For advanced users only,
this can be edited to create your own custom setup.

3) Restart Creo

*** Notes:

1) To configure your installation using the supplied .bat files, you
must have read/write access to the your
   Creo loadpoint folder (and subfolders).  This folder is typically
"PTC".

   In Windows 7 and Vista, this typically requires you to disable the
UAC (User Account Control) and restart your computer.

2) Prior to running a configuration .bat file, you will see drawing
templates for BOTH MMKS-ASME (A1,A2,A3,etc)
   and INLBS-ASME (A,B,C,etc) drawing templates.  You should only use
the templates that are relevant to your
   unit systems.

3) The following folders contain drawing standard specific Drawing
Templates:

     templates_asme_in – Inch length unit, ASME standard drawing
templates, Sizes A,B,C&D.
     templates_asme_mm – MM length unit, ASME standard drawing
templates, Sizes A1,A2,A3,A4.
     templates_bs_mm – MM length unit, BS8888 standard drawing
templates, Sizes A1,A2,A3,A4.
     templates_iso_mm – MM length unit, ISO standard drawing
templates, Sizes A1,A2,A3,A4.(1st ANGLE PROJECTION).
```

In my case I have activated the one in bold above

2. **Double Left-Click** the appropriate batch file.

3. Now when you start the program the relevant standard is set.

If you want to use the default set of Formats and Templates then this is all you need do to set things up. However if you notice some anomalies in your drawings then you may have to edit the *.dtl file, to do so see Appendix 1.



Above is an example of a Drawing Setup File (*.dtl).

About Drawings

Using the Creo[tm] Parametric Drawing mode, you can create drawings of all models, or import drawing files from other systems. You can annotate the drawing with notes, manipulate the dimensions, and use layers to manage the display of different items. All views in the drawing are associative: if you change a dimensional value in one view, Creo[tm] Parametric updates other drawing views accordingly[1]. Moreover, Creo[tm] Parametric associates drawings with their parent models. The model automatically reflects any dimensional changes that you make to a drawing. In addition, corresponding drawings also reflect any changes that you make to a model (such as the addition or deletion of features and dimensional changes) in Part, Sheetmetal, Assembly, or Manufacturing modes.

[1] Only those parametric dimensions used to create the 3D model can do this. Extra dimensions added manually cannot.

Detailed Drawings Module

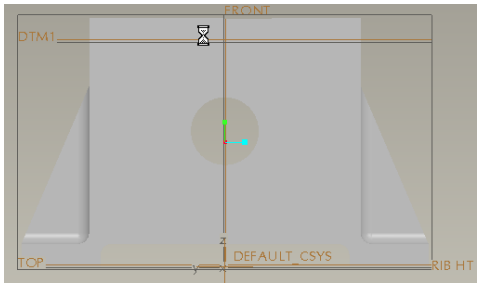
Detailed Drawings extends the drawing capability of Drawing mode. You can use it with Creo[tm] Parametric to create, view, and annotate models and drawings.

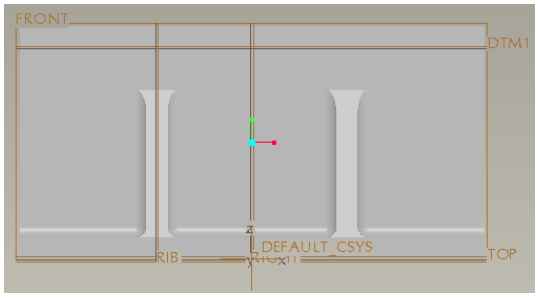
Detailed Drawings support additional view types and multisheets, offers numerous commands for manipulating items in a drawing, and lets you add and modify different kinds of textual and symbolic information. In addition, you can use it to customize engineering drawings with sketched geometry, create custom drawing formats, and make multiple cosmetic changes to drawings.

With Detailed Drawings, you can also use a shortcut menu to modify any object in a drawing from anywhere in the model tree. At any time when a drawing window is active, you can interrupt your current process and activate a drawing object for modification.

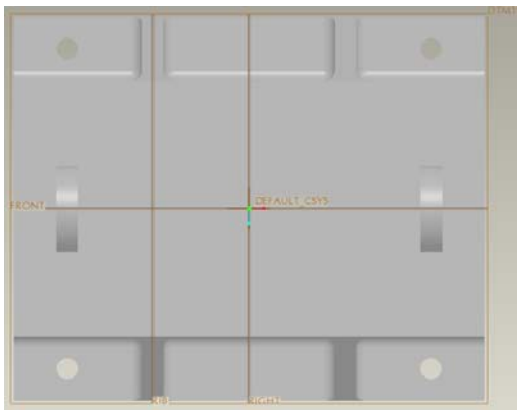
With Creo[tm] Parametric Interface or Detailed Drawings, you can access various interface commands for exporting drawing files to other systems and importing files into drawing mode.

To understand the views you can get from your model or assembly, in your 3D model, use the Saved Views; here you can set Front, Top, Left, Right, Trimetric and Isometric views plus a few others. Try it now, in the modelling mode; alter the model to **No Hidden, (Left-Click**) as you do so. You will soon see how Orthographic and 3D views are related in Creo™ Parametric





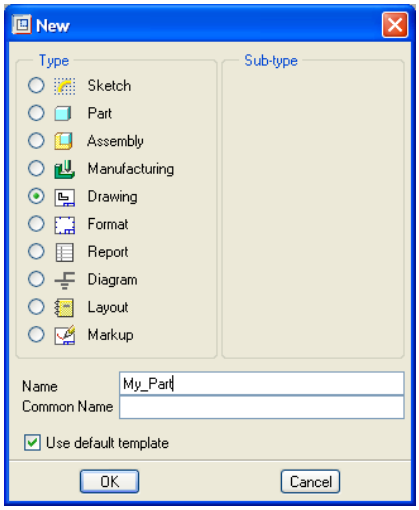
Above is the **LEFT** view, to the right is the **FRONT** view and below is the **TOP** view. Note how one relates to the other. The datum planes on show should help you understand this.

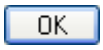


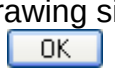
Create a New Drawing

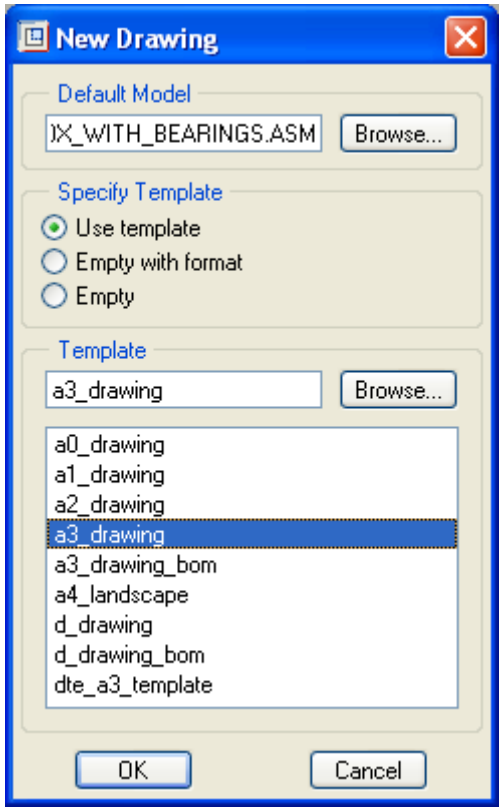
With a 3D model open on the screen

1. **Left-Click File** then click **New**
2. Select the **'Drawing'** radio button
3. Type in a name for your drawing file



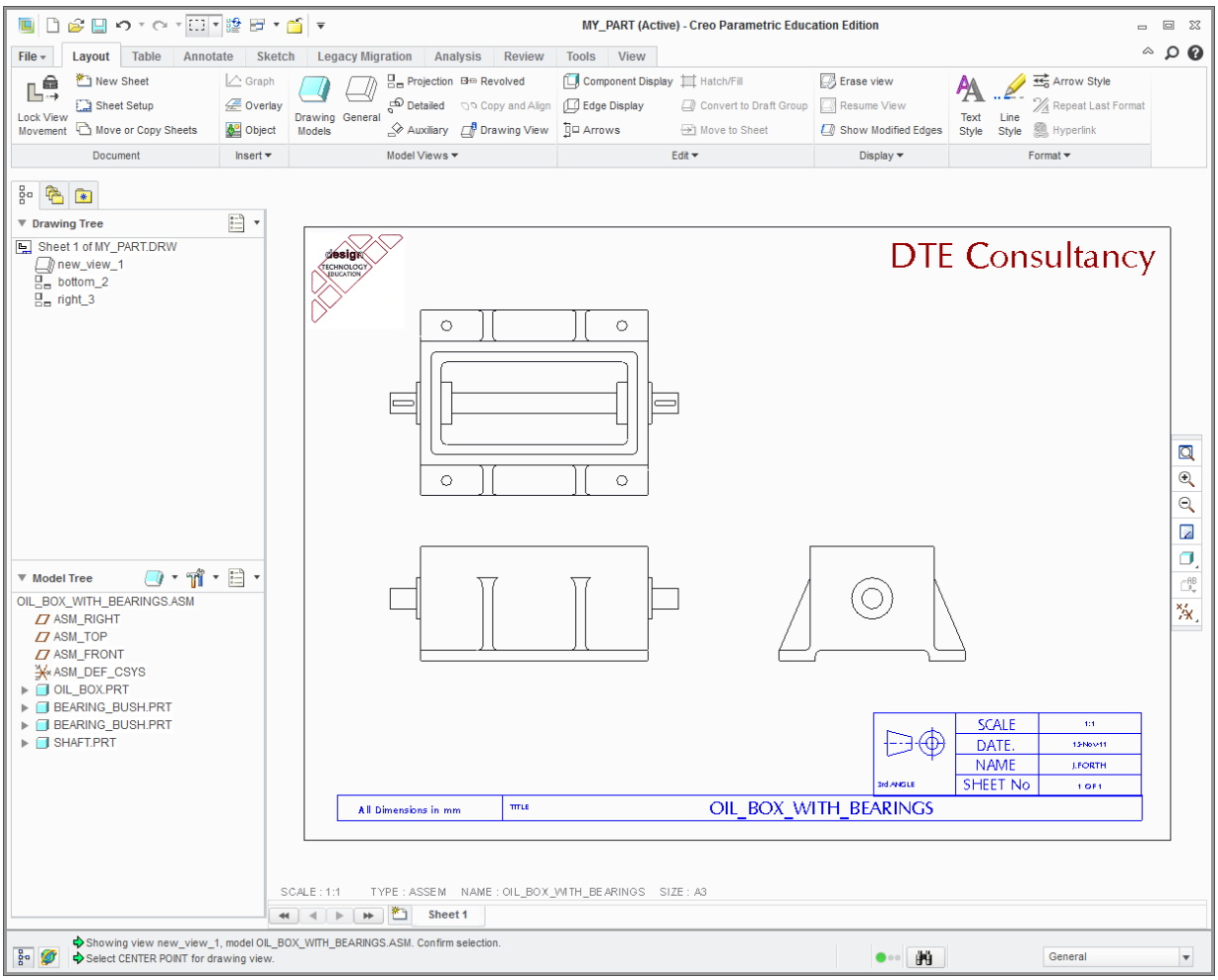
4. Make sure the 'Use default template' box is ticked.
5. **Left-Click** 

6. Pick the drawing size you want and **Left-Click** 



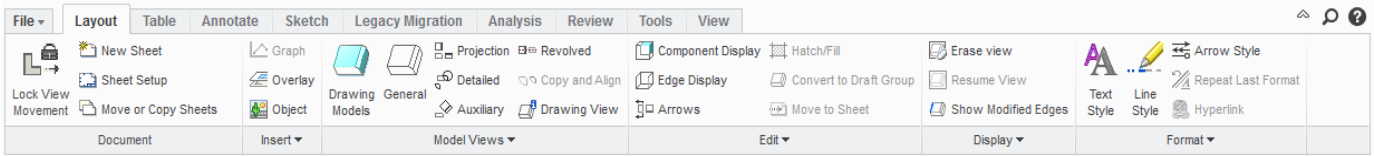
Depending on the part or assembly you have open on the screen; the design of the format and template you use, you will now get the 2D drafted representation you require. It will not be complete but of that later.

(Note: If you have Datum planes *and Coordinate systems* *open to view then they will appear in the drawing. Just turn them off and* ***Left-Click*** *anywhere in the Graphics window to see them disappear or* ***Left-Click*** *repaint* *or Ctrl+R)*



If you look at the picture above you will see what is meant. There are certain elements missing; i.e. dimensions and some elements may be wrong for your standard; the representation of the filleted corners for example. Note the blue line (pre-selection highlight) around the view that will help us change things. Note also such things as the border around the edge, the folding guides and reference numbers, the text boxes etc. This is known as the **Format** and you can create your own format. The three views haven't appeared by magic. Their position and appearance is due to another file that overlays the format and is called the **Template**. It is the combination of your model, the format and template that makes the drawing unique (see diagram on the next page).

First take a close look at the screen shot and note that it has a new feature, this is called the Ribbon Interface.



Each tab on the ribbon allows different options into play. Left-Click each tab and note the Commands that appear that also appear in other ribbons. Right-Click in the graphics are and note the commands available for rapid use.

Select each drawing tab to observe the functionality found in each:
• **Layout** – Sheets, formats, views, display settings and drawing objects are controlled from this tab.
• **Table** – Tables are created and edited using tools in this tab.
• **Annotate** – Dimensions, notes and tolerances are shown, created and controlled using tools in this tab.
• **Sketch** – 2D draft entities can be sketched using tools in this tab.

- **Legacy Migration** – Take old models and those from other software
- **Analysis** – Similar to the analysis ribbon in the modeling mode.

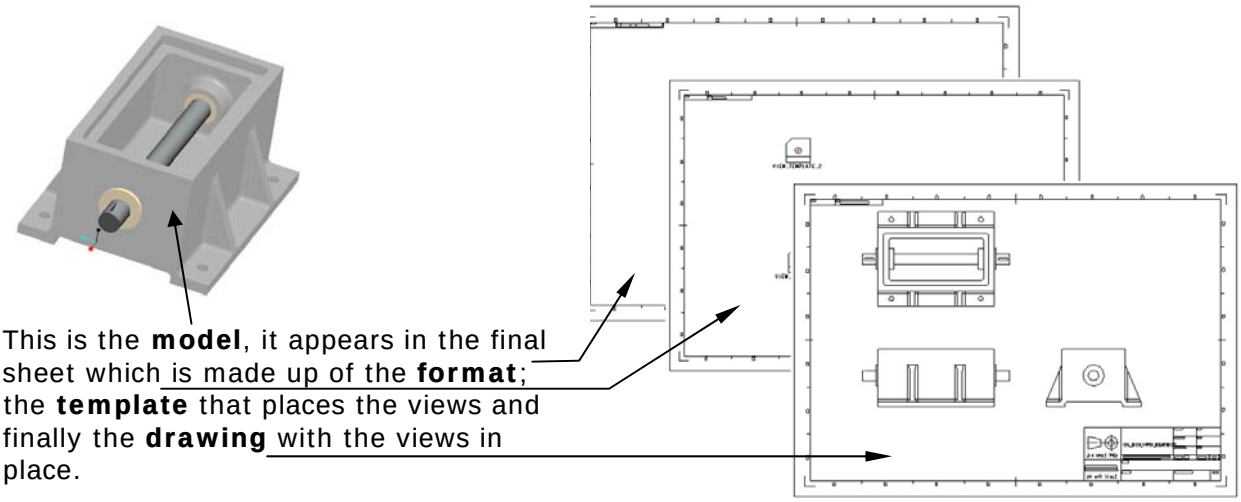
• **Review** – Update your drawing, compare different versions, query for information and take measurements using tools in this tab.

- **Tools** – Various drafting tool and access to the template creation mode.

• **View** – Tools to change the view parameters.

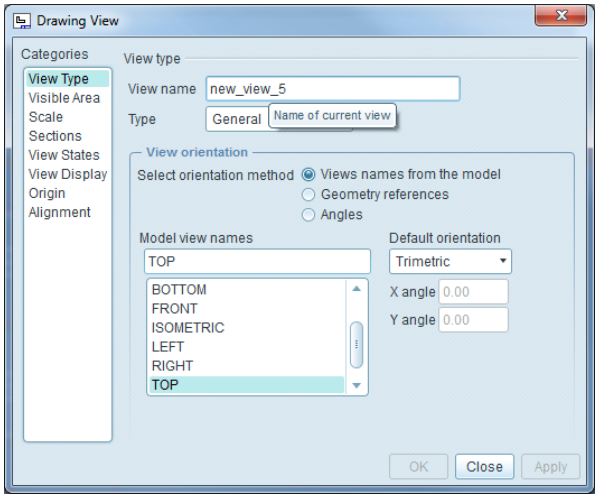
As we progress through the tutorial these features will be addressed in more detail.

Relationship of model and drawings



This is the **model**, it appears in the final sheet which is made up of the **format**; the **template** that places the views and finally the **drawing** with the views in place.

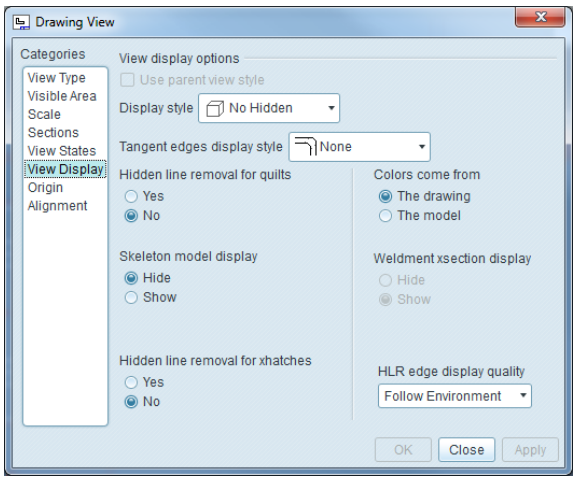
7. Double **Left-Click** on the blue line, it goes red indicating selection, the following dialogue box appears, the use of this will allow you to modify the views.



Note: The list of categories down the left hand side.

8. **Left-Click** each and see what you get.
9. You can play about with the options and if you **Left-Click** the Apply button you can see how the view changes; if you don't like it change it back.

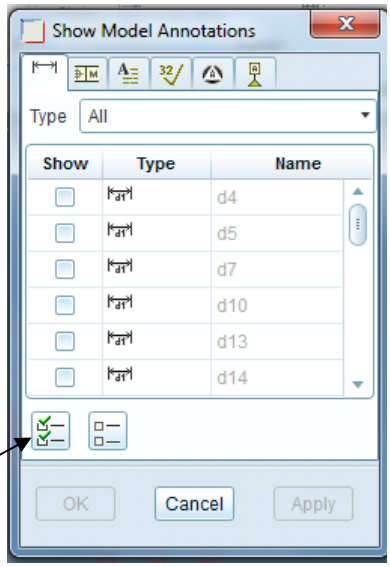
The important one at this stage is the **View Display**. Here we can show or hide Hidden lines and remove tangent edge display. On our own templates, created later, we can automatically set these functions.



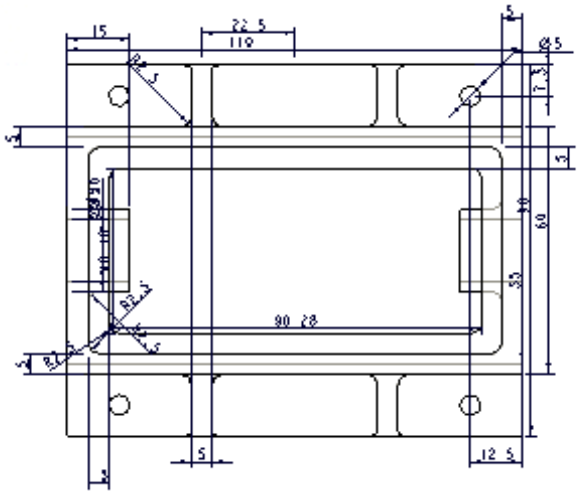
10. If you want to show hidden lines and no tangent edges then choose as shown here.
11. Once you have finished **Left-Click** OK or Apply then Close

Adding Dimensions

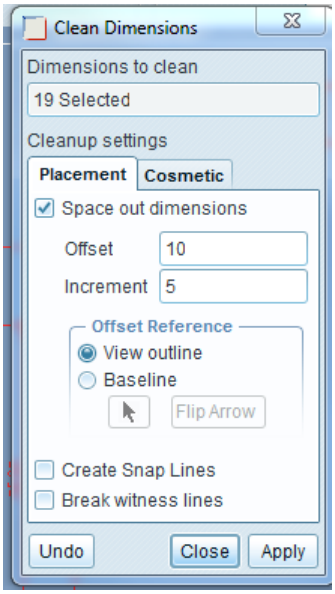
1. Select the view that you want to add dimensions. **Left-Click** the Annotations Tab in the Ribbon Inteface.Left-Click the blue line, you will find it by floating the mouse near to the view.
2. **Left-Click** Show Model Annotations and the following floating menu will appear.



3. **Left-Click Show All Dimensions**. Don't panic, I know there are a lot and they are in the wrong places. We will proceed to edit their position and remove the unwanted.
4. **Left-Click** OK
5. If you select the view again and **Right-Click;** In the Floating Menu you will see the option to **Cleanup Dimensions**. This allows you to quickly move things about so you can see the wood from the trees.

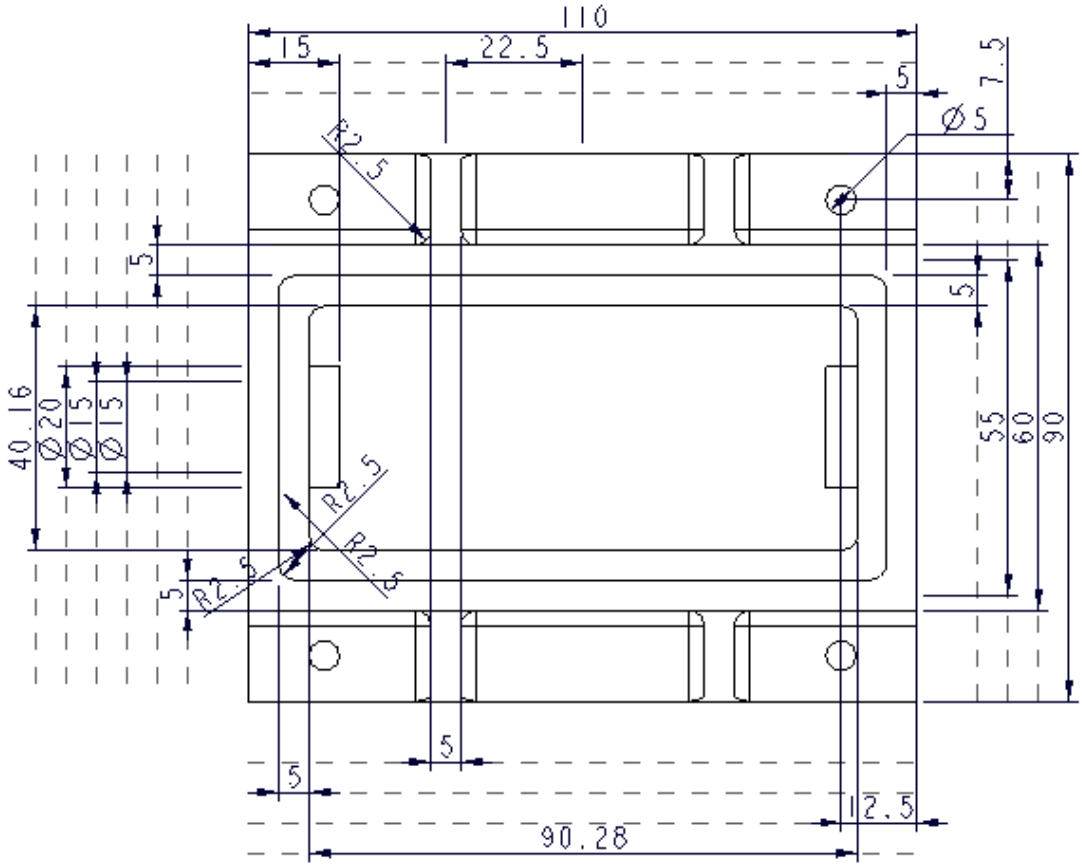


The Offset is for the first dimension and defaults to 12.5mm, each subsequent (Increment) dimension is 5mm on from the offset. A set of dotted guide lines is shown to help set them up. These can be deleted as explained below. To not show them uncheck the **Create Snap Lines** box



For the moment just accept the defaults

6. **Left-Click** Apply Close



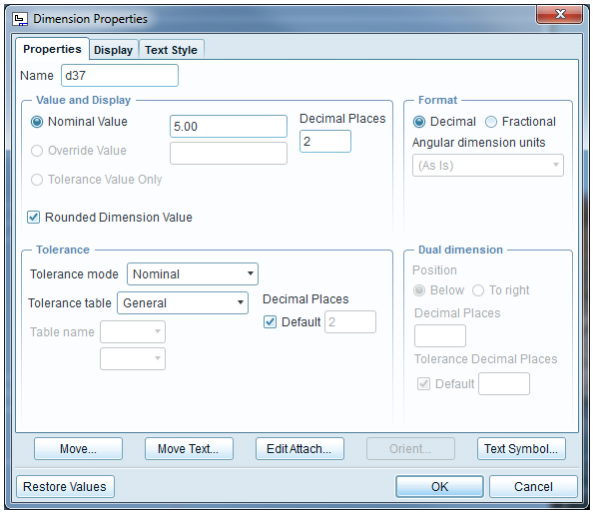
7. Now **Left-click** to select and Delete all the hatched guide lines; (Use the **Ctrl** key to select multiple lines and then hit the **Delete** Key (see drawing above)

8. Now select those dimensions you want to delete, either one at a time or multiple select using the **Ctrl** Key select with a **Left-Click** then **Right-Click** and select **Erase**. (If you hit the Delete key they will disappear immediately)

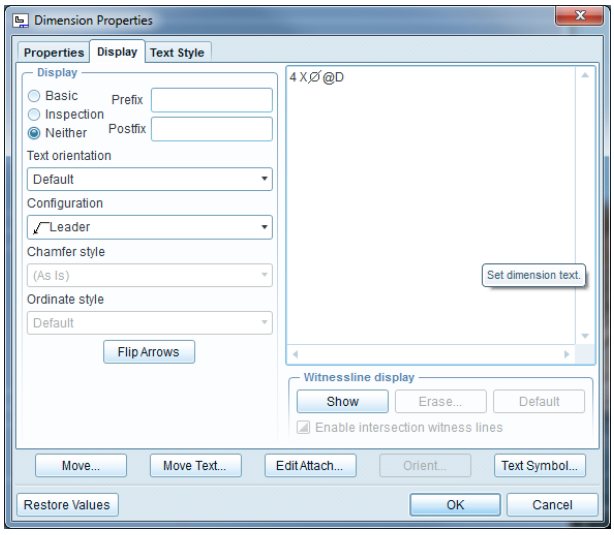
The dimension(s) will grey out and only disappear when you select the next one or anywhere in the Graphics Window. This gives you the opportunity to **Right-Click** and **Unerase** if you accidentally choose the wrong one. Whilst you are doing this you can also **Left-Click**-**Drag** to reposition the dimension and/or the value.

9. Spend some time 'tidying up'.
10. In Creo^{tm} Parametricyou can move a dimension from one view to another. The procedure is very simple.
    a. Select the dimension to move with a **Left-Click**
    b. **Right-Click** and select **Move item to view**
    c. Select the view to receive the dimension. Note: It has to be one that has the feature orientation that is suitable.

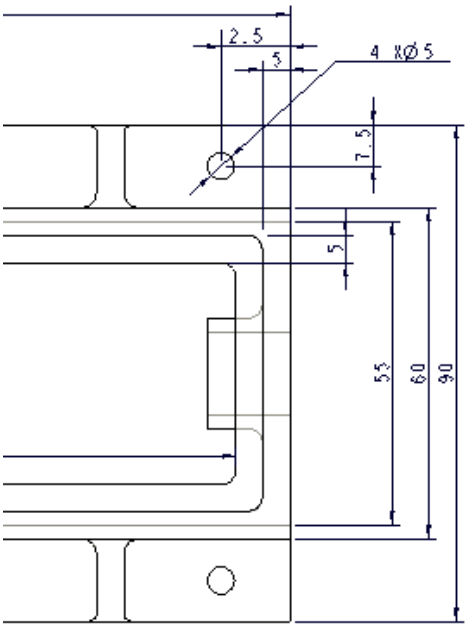
11. You can add a note to a dimension value by Double **Left-Click** the value and follow the instructions in the dialogue that appears. As shown here

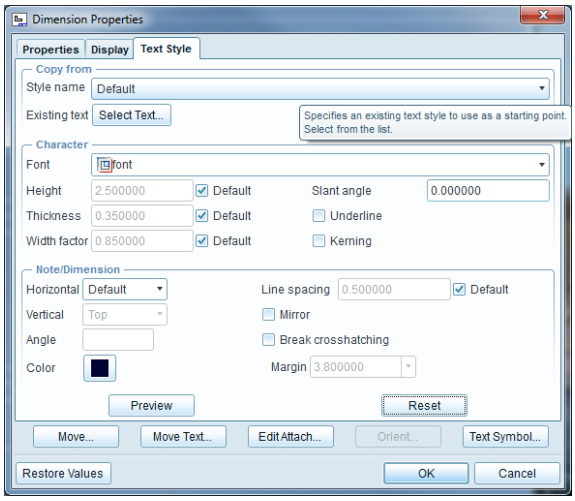


You can alter a lot of things on these three boxes to determine exactly what you want. In this example we have added the prefix "4 X" to the Ø@D. (This is the code for the Ø5mm of the four mounting holes.) This is neater than dimensioning each one.

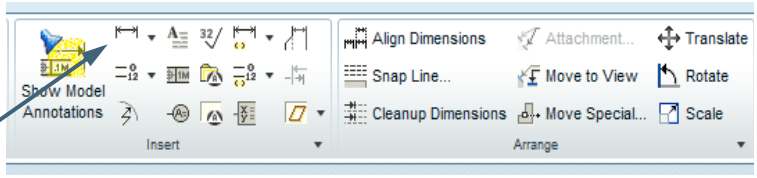


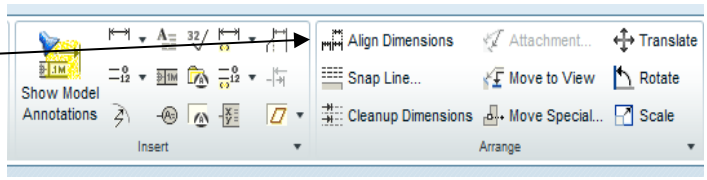
Your view will now look a bit like this.



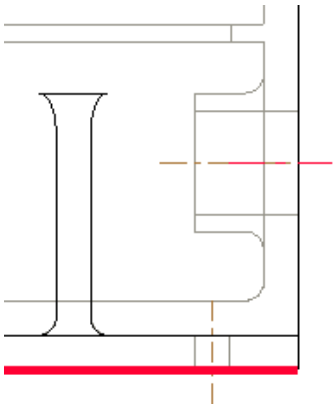


12. You can now manually add dimension using the same technique as in the Sketcher.
13. Select the **Create Standard Dimension** Tool from the Toolbar
14. Also **Align Dimensions.** First selected will be the datum. Other will align to it.

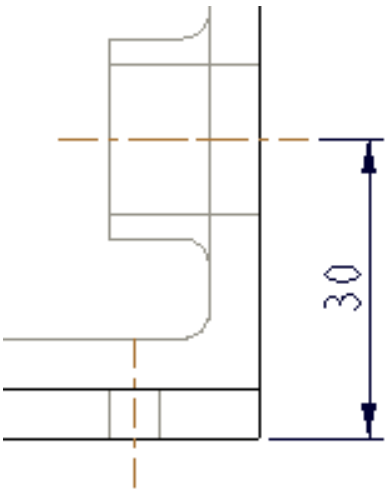




15. **Left-Click** to select the first object of the dimension; edge, axis, etc., **Left-Click** the second object.

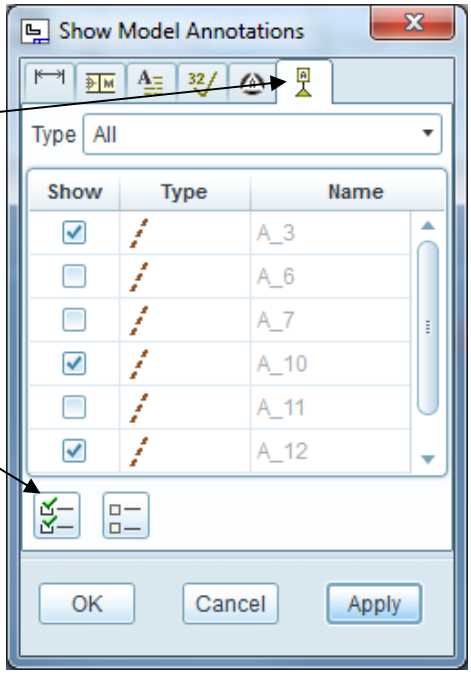


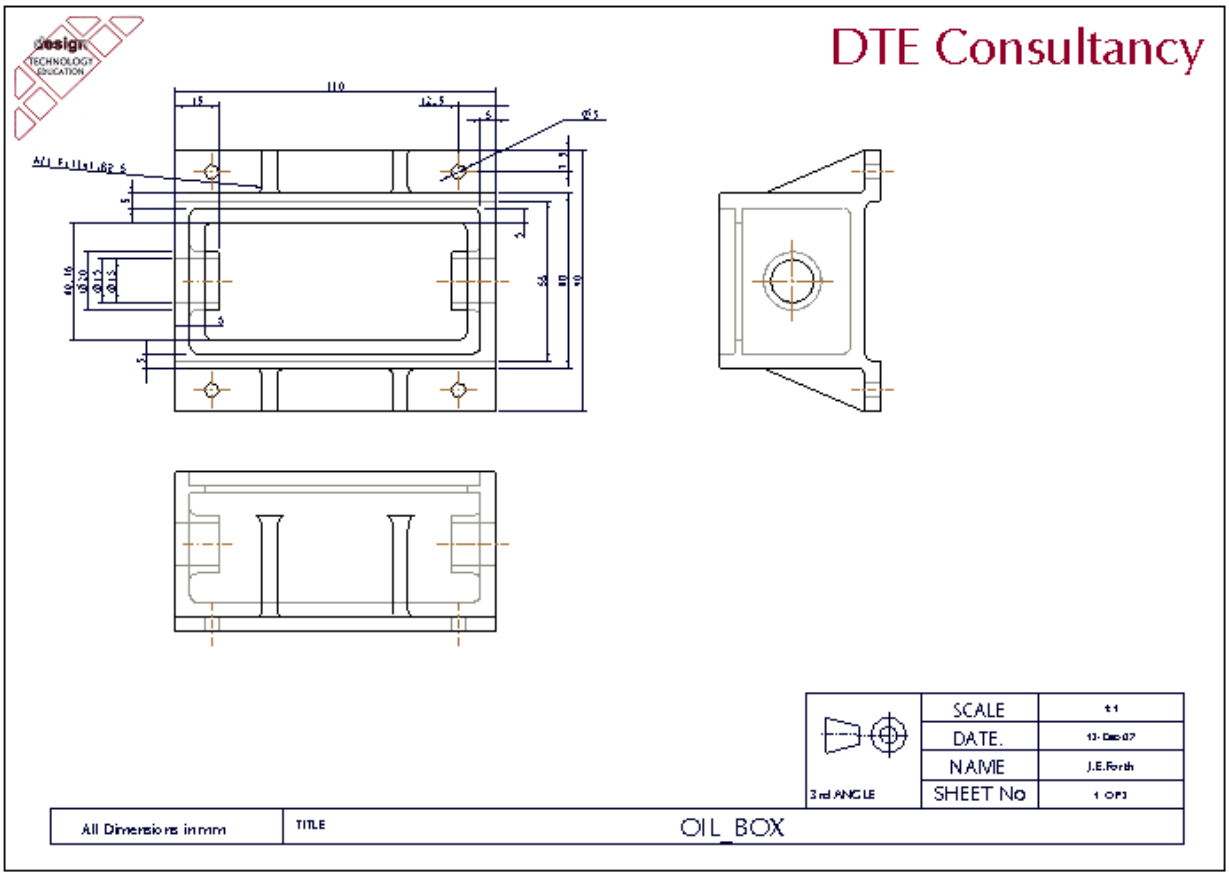
16. Then **Middle-Click** where you want the dimension placed.



Note: These added dimensions are not parametric and will not change the model if you try to change them in the drawing.

17. To add Centre Lines (Axis) Left-Click the view on which they are to be placed.
18. Left-Click and select the Axis Tab.
19. Put a check in the box of the Axis you want, or select all.
20. **Left-Click**



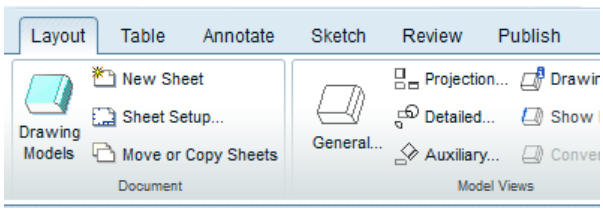


Notes can be added as can a Bill of Material (BOM) table and much more. If you want to use this function see the section **To Add a Bill of Materials (BOM) Table.**

Remember that these apparent drawings are in fact representations of 3D objects and assemblies. As such any changes made to them will be reflected in these drawings. This Associativity is an important productivity aspect of **Creo™ Parametric**. You can try it by opening the part,

1. (**Right-Click** in the Navigation window on the part and choose **Open**),
2. Change a feature of the part or add an extra chamfer. Now go back to the drawing and see that the change has already been made.
3. In some cases you may need to regenerate. **Left-Click** the drawing. This is the essential difference between a 3D modelling CAD and a 2D drafting CAD.

Adding new sheets



You can add sheets to the file.

1. **Left-Click New Sheet** on the Layout Tab. This will come in as a blank format onto which you can place general and projected views, create sections or even exploded views.

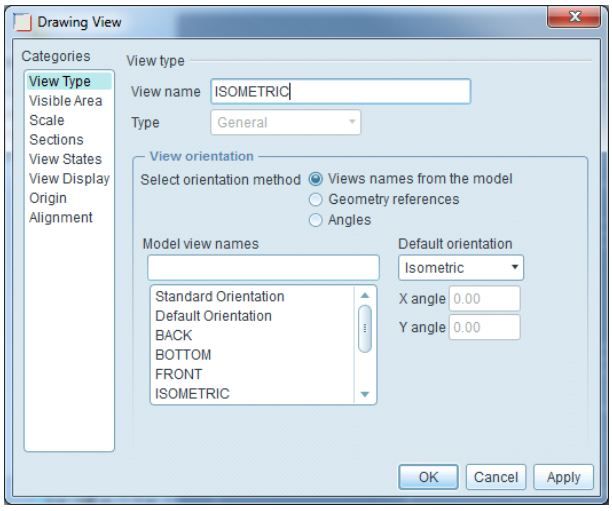
Pictorial Views

When you create your part or assembly you also can create views using the **View Manager**. Also with **View Manager** you have the ability to set special 3D views that best show your model. These views can also be used within drawings.

To create a special view move the model in the Graphics Window as required.

1. Open the **View Manager**
2. **Left-Click**<**Orient** ,
3. **Left-Click<New** and type a name e.g.3D.
4. Close the **View Manager.** Return to the Drawing window
5. On the new blank sheet **Right-Click** and select from the menu **Insert General View**.
6. **Left-Click** on the sheet, where you want to place the view. The drawing view dialogue box opens to allow you to finalise the view.

It opens in **View Type**.



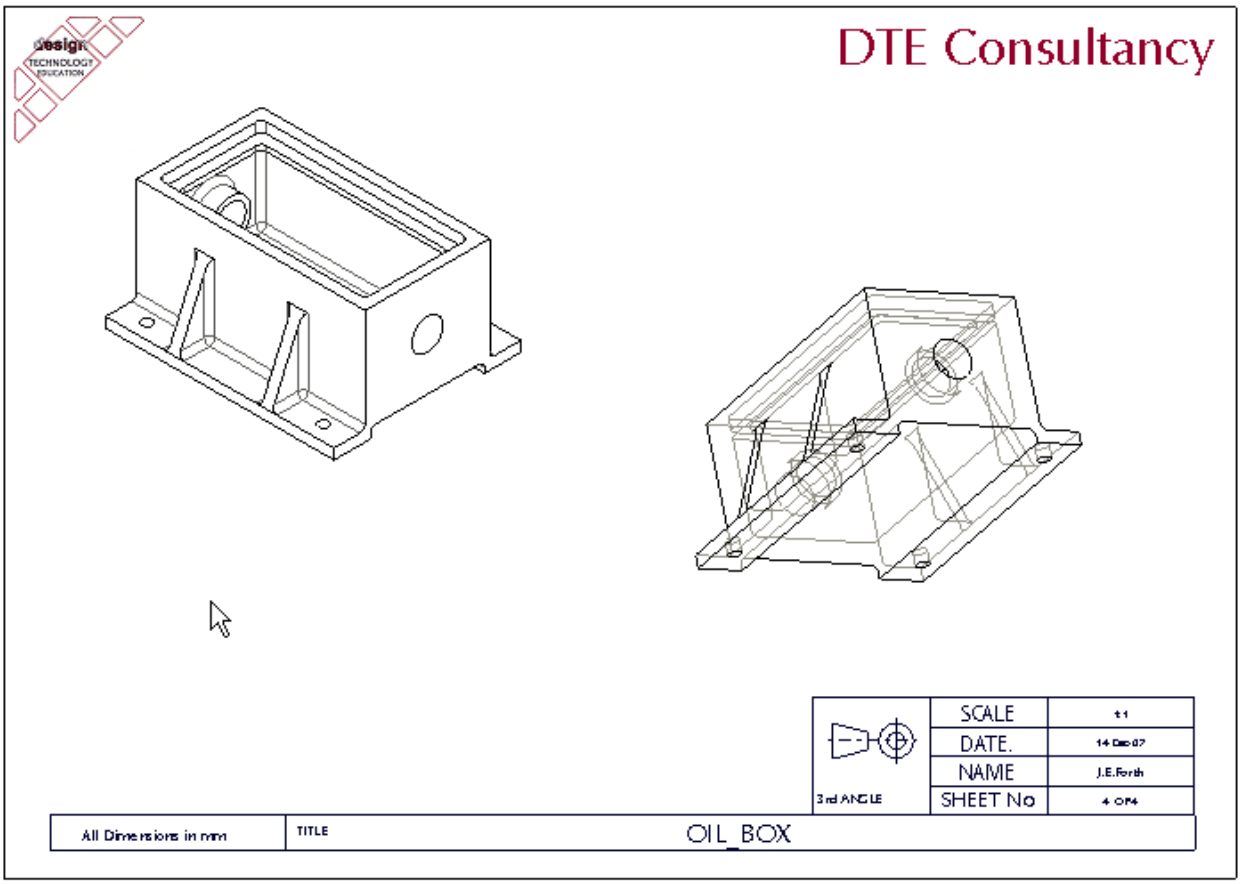
1. Type in a model name and then scroll down the list of **Model View Names** until you find the one you want. Trimetric, Isometric and your own 3D views are all available.

2. **Left-Click** Apply; notice the view is shaded as it appears in the part modeller.

Use other functions in the Drawing View dialogue to set other aspects of the view as you desire, in the same way as described previously. To set a different scale go to the **Scale** category and select **Custom Scale**, place a decimal number to represent your needs, e.g. 2.0 will double the scale from the default.

On completion **Left-Click** Apply then Close.

You will notice that the view is locked in place. If you want to re-position the view on the sheet you will need to select the view, (Red line around it); **Right-Click** and un-tick **Lock View Movement** then **Left-Click-Drag** to the new position. To lock the view again, reverse the procedure. You can place more than one view on the same sheet.



In this case the right hand view is a specially created one called 3D and it is shown with Hidden detail showing. These appear as greyed out lines but will print-out or plot using whatever standard you have set; they appear as dashed lines in BS8888 for example.

Creating a sectional view

To create a sectional view you have to start in the part /assembly file. The simplest way of creating a section is to make sure there is a datum plane lying just where you want the section to be. You can either use the ones created when the model was made or you can place a plane with its sole purpose being to create a sectional view.

You can also create a sectional view within the drawing file itself; see the section headed ‘Sectional View Created in the Drawing file’. Creating the sectional view within the model does allow you to see the finished section and also use it to create a 3D sectional view, something unique to 3D Modelling packages.

1. Activate the **View Manager** and select the Xsec tab.
2. **Left-Click New**; type a name or letter and hit return. You will be presented with some choices in the Menu Manager.

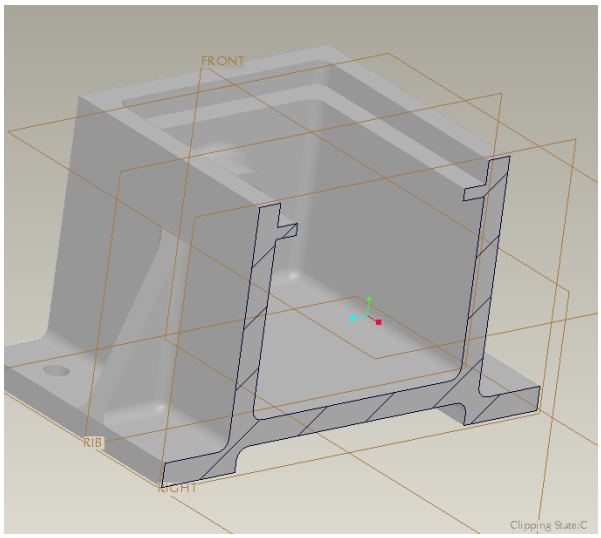
3. Accept the defaults and **Left-Click Done.** The **Menu Manager** changes and you have to create or select the **Datum Plane** you intend to use as the cutting plane.

4. In the Graphics window select the plane required. You will be then back in the **View Manager**.
5. **Left-Click Display**, Select **Set Active** and the model will be sectioned to show how it will appear. If you want to cut off the opposite side **Left-Click Display Flip**.

The view will appear as below. To return to normal view

Double **Left-Click No Cross Section.**

You can go on and create as many different sections as you think you require. When complete Close the **View Manager**.



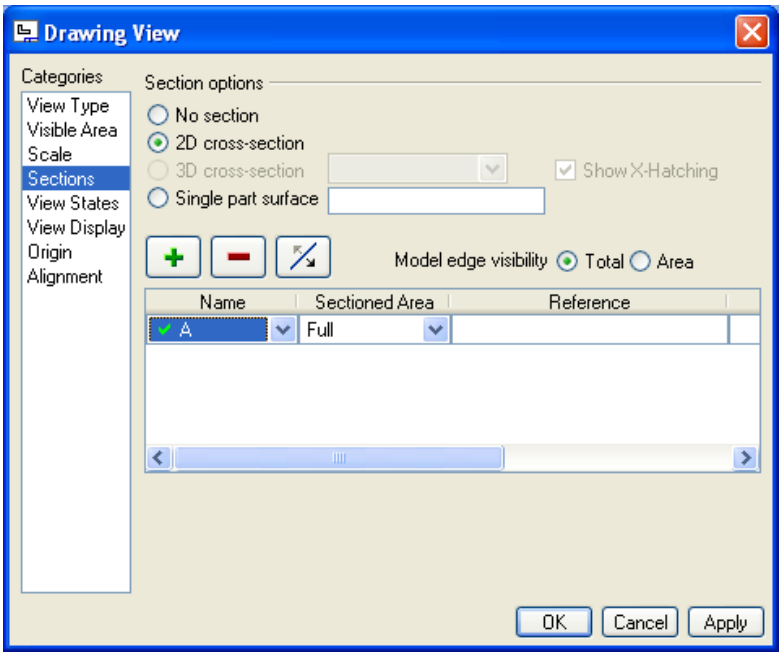
You can now go to the drawing view and create the sectional views you require.

6. Create a general or projected view on the sheet, this view will be sectioned. To create a projected view. Select the parent view; **Right-Click** and select **Insert Projected View**. A ghost of the view will appear **Left-Click** when you have positioned it. Whether it is a 3rd Angle or 1st Angle will depend on how you have set up the Drawing Options File (see Appendix 1).

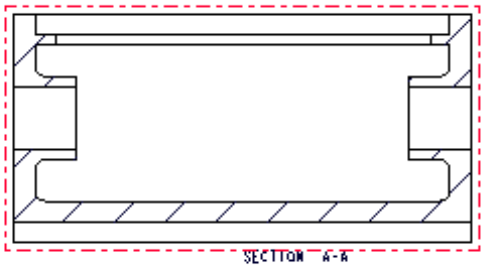
A.

a. Select the view with a Double **Left-Click** to open the **Drawing View** box.

b. Select the **Sections** Category



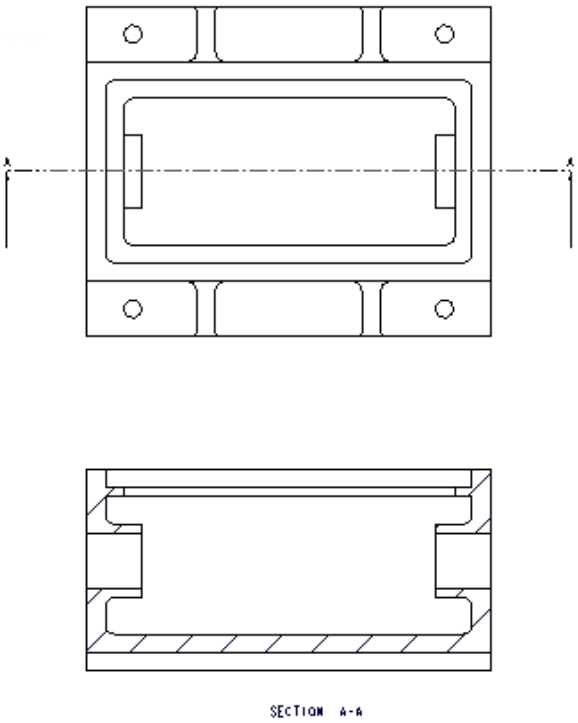
7. **Left-Click** select the radio button 2D cross-section
8. **Left-Click** the "+" and the available Xsec will appear. Note: Some will obviously not be possible on some views.
9. **Left-Click** to select the one you want and **Left-Click** Apply. The view will change to appear as below.



Left-Click Close

Select the view and **Right-Click** select **Add Arrows** and then choose the view to which the arrows should be attached.

The view will now look something like this, see below.



If the hatching is too widely spaced or at the wrong angle Double **Left-Click** on the selected hatching and a Menu Manager will appear that will let you change whatever is needed to create the right effect. The Section title can be repositioned by **Left-Click-Drag**.

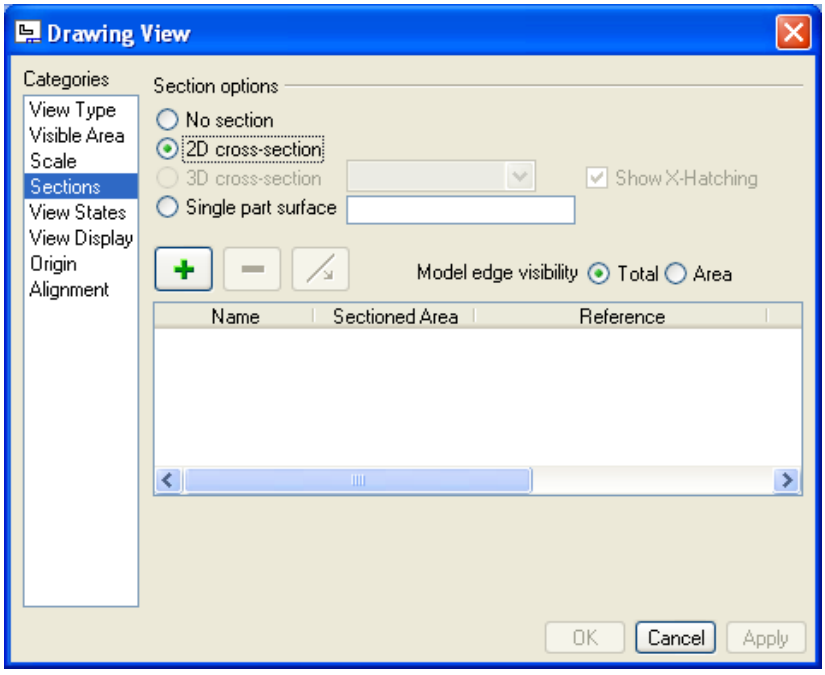
To remove hatching from any part select it by Double **Left-Click** and set the spacing so wide that it disappears.

Sectional View Created in the Drawing file

If you have not created the sections within the model file it is still possible to create the section from within the drawing window. Ensure you have the Datum Planes visible and a **Projected** view created that will be sectioned.

a. Select the view with a Double **Left-Click** to open the **Drawing View** box.

b. Select the **Sections** Category



At this point there will be no Xsec in the box.

1**. Left-Click** '+' and Create New

2. Select Planar<Done

3. Enter a Name or Letter

4. In the Menu Manager **Plane** will be highlighted if not select it.

5. In the graphic Window **Left-Click** the Datum plane to be used (in an assembly use a ASM datum plane, you may need to **Right-Click** and use **Pick from list** to find the required plane) Or select from the Model Tree

6. In **Arrow Display** select the view in which the arrows will be displayed.

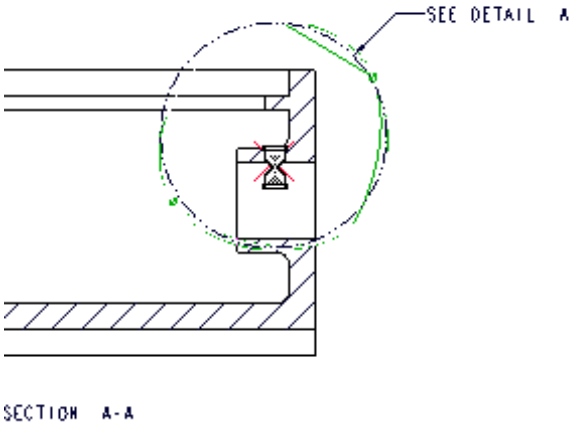
7. **Left-Click** OK

Enlarged Details

To create a detail is quite easy. It is in three parts, first select the feature you want to highlight; then create a spline curve surrounding the part; then place the detail view. If a sectional detail is selected then the detail will be a section.

1. **Left-Click Insert<Drawing View<Detailed**

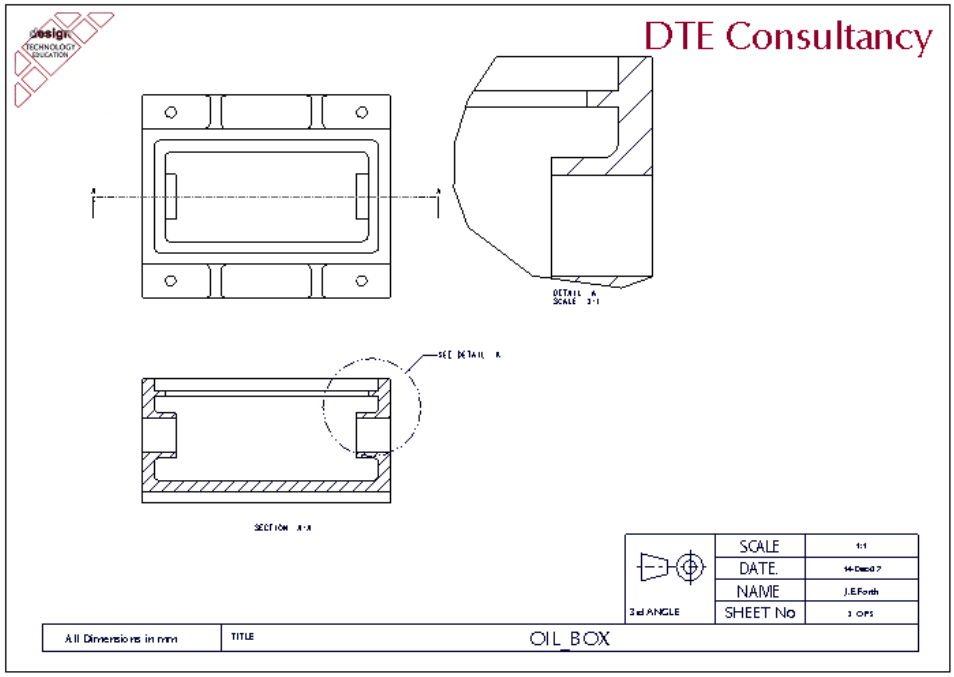
2. **Left-Click** to select the feature, edge or face etc. A cross will appear that indicates the centre of the detail.



Move away from the centre cross and create a spline curve surrounding the detail. (There is no need nor is it possible to use the Sketch Spline tool)

3. You do not have to complete the curve, when you get close **Middle-Click** and a dashed circle will appear.

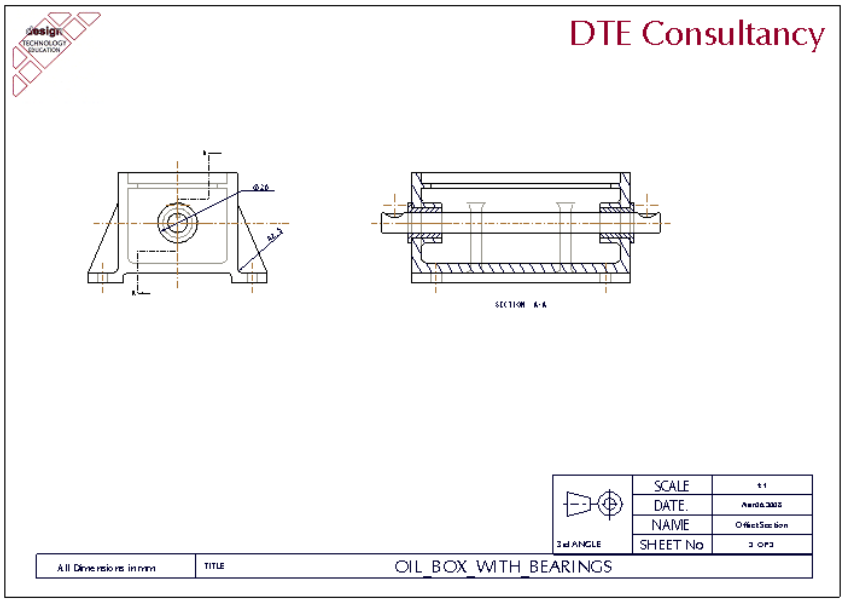
4. **Left-Click** in the position you want the detailed view and it will appear. The default scale is 2:1(set in the *.dtl file) but **Double Left-Click** the scale value to enable it to be varied.



The sheet will now look like this. Dimensions can be added to detailed views.

Offset or Stepped Section

There is occasionally the need to create a sectional view that uses a set of offset lines or steps to define the section cutting plane. To do this is not difficult but requires a clear approach to the objective.



Here is an example of a Stepped or Offset Section, Note that lines denoting the steps are not shown in the sectioned view, however since this is an assembly the hatchings are separate.

Follow the instructions to create a Stepped section

1. Insert a general view to be the parent of the Stepped Section or choose an existing view.
2. **Right-Click** and from the floating menu select **Insert Projection View** and place it.
3. **Double Left-Click** the projected view and choose **Sections** from the **Categories** list in the **Drawings View** dialogue.
4. Choose **2D section**
5. Choose
6. Choose **Offset < Bothsides**
7. Choose Done and enter a letter in the dashboard area to label the section, hit **Enter**

Your sheet should look like this.

What happens now is that a new window opens that is very much like the sketcher. Notice the face on which we are to place the stepped line has been selected.

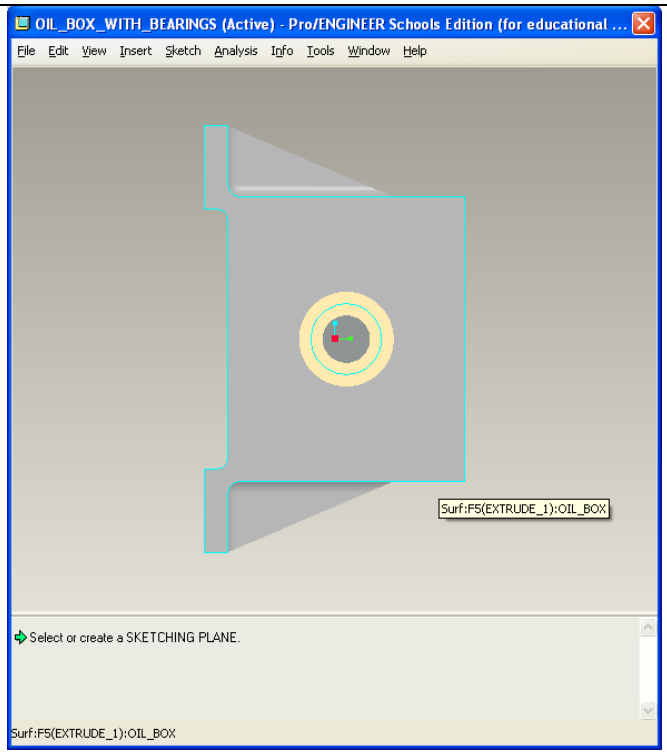
Select the face and accept all the defaults.

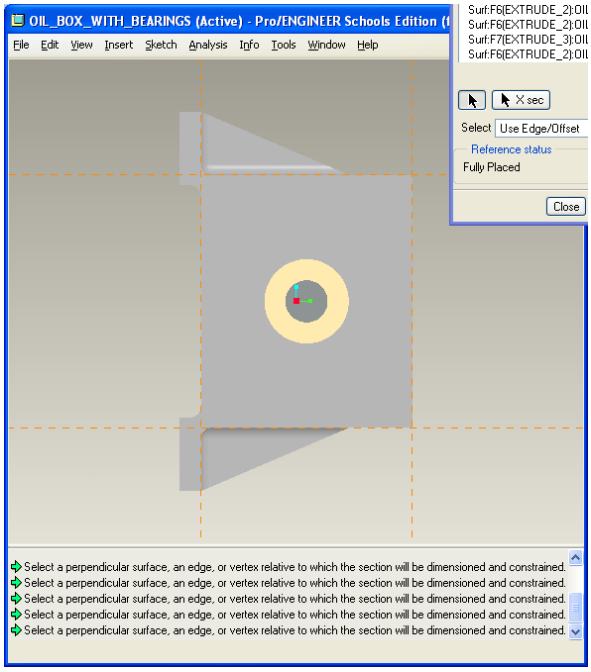
8. A References window opens

.

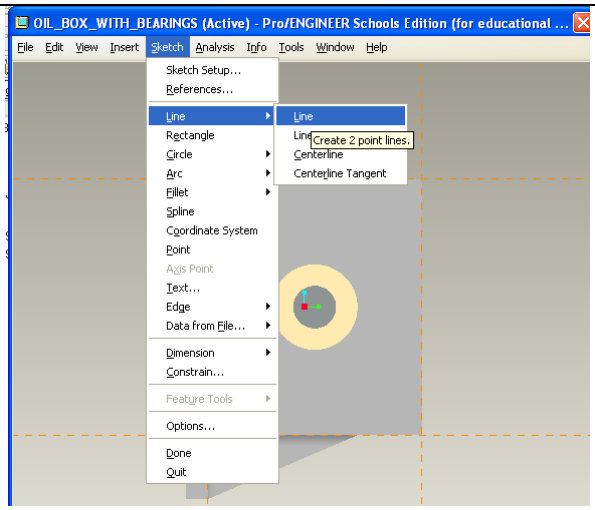
9. **Left-Click** to select 4 edges to enclose the area on which to sketch the stepped line as shown opposite. **Close** the reference dialogue.

10. **Left-Click Sketch < Line<Line**

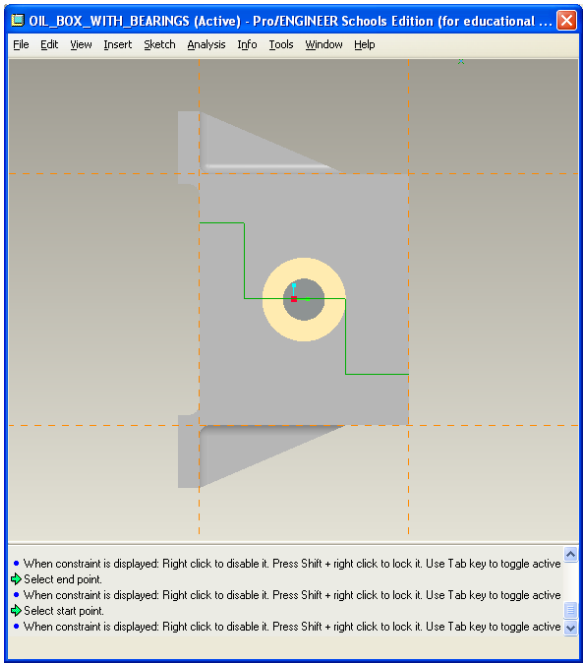




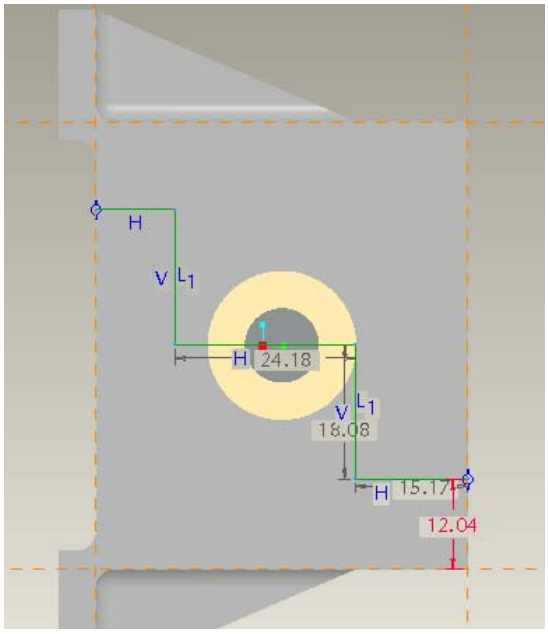
11. Sketch the stepped line as if it where in the sketcher starting at one of the reference edges and finishing on another reference edge. Use the automatic Horizontal and Vertical Constraints, there can be no lines at any other angle.



12. This line can be dimensioned in the same way as in the sketcher. Select **Sketch <Dimension <Normal.** Use the Constraints tool to align with axis and any other feature necessary



13. To complete the line creation choose **Sketch < Done**

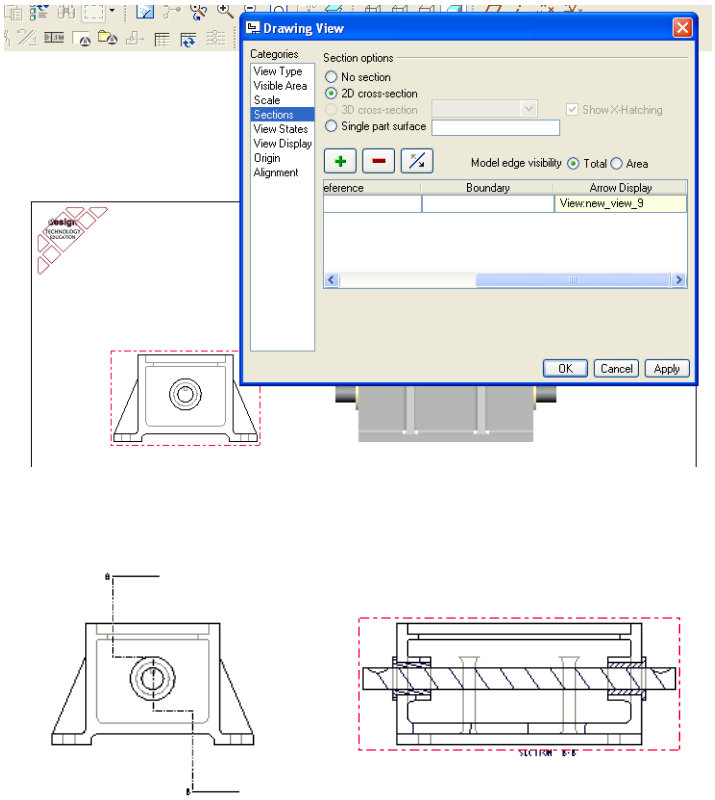


14. Select the view for the arrow

display as shown opposite.

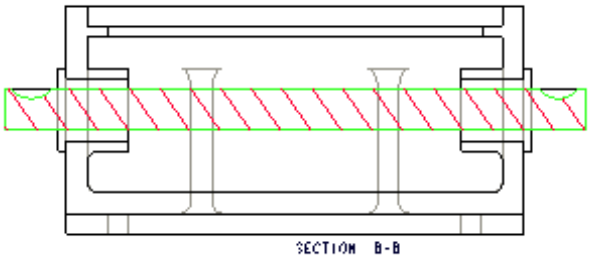
15. Choose **View Display** to modify the view as required to show hidden line and tangent edges etc., **Left-Click OK**

16. The stepped sectional view now appears and can be modified as required. Note: The hatching of the shaft is not to BS8888 and therefore needs to be removed. The next section deals with modifying the hatching lines



Modifying Cross-Hatching display

Double Left-Click on any of the hatching lines in the view. This will bring up a Menu Manager with a number of options.



The top part of the box allows a choice of options about what to modify and the bottom part where to modify.

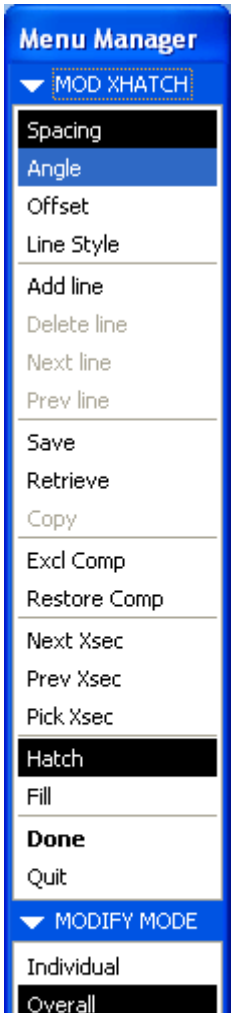
Changes can be made to the spacing, the angle, the offset or the line style.

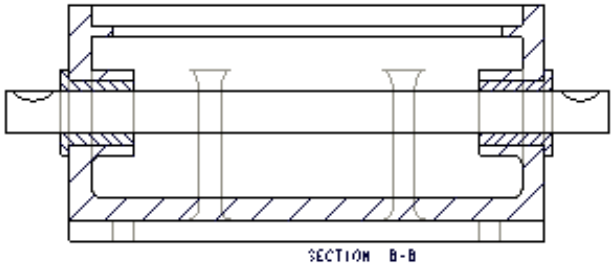
Since this is an assembly there are

more than one section that is hatched.

Changes can be made to all the hatchings (**Overall**) or each one separately (**Individual**). You can choose which one on which operate either in sequence (**Next Xsec; Previous Xsec**) or by selection (**Pick Xsec**).

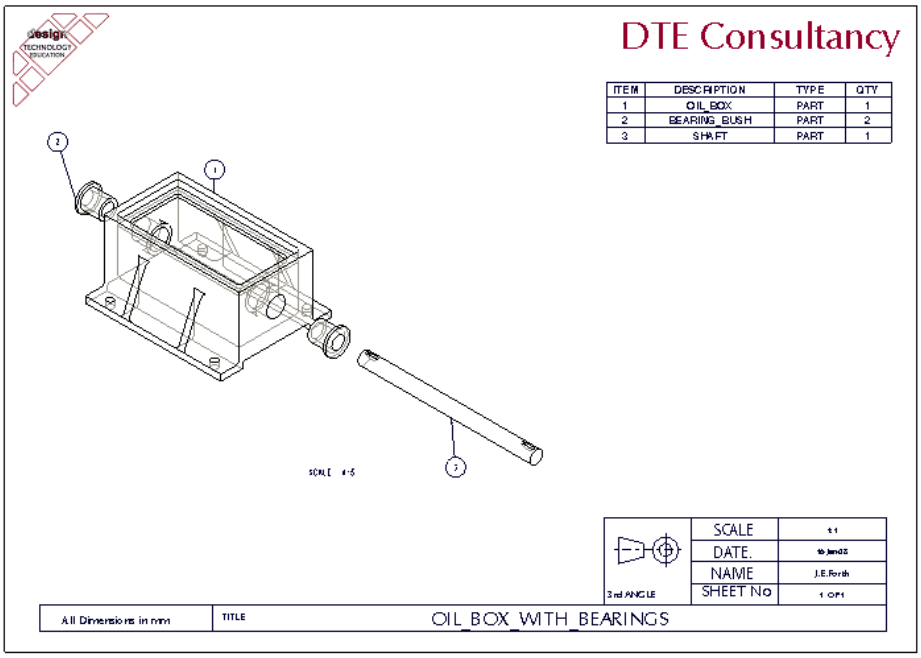
1. Select the Xsec to modify
2. Select the value e.g. Spacing
3. Select Half or Double or Value (Note: to remove a hatching select Value and state an absurdly large value e.g. 2000mm in the dashboard)
4. Select another value to modify or move on.
5. Choose **Next Xsec** to move to the next in sequence to modify.
6. When all modifications are complete choose **Done** and the view will reflect those changes.





Exploded Views

The exploded view has to be created via the View Manager within the assembly file. The details of how this done is the subject of another tutorial. Once the exploded view is created it can be placed in the same way as described for pictorial views.



An example of an exploded view with a BOM table and balloons.

Other Views

It is possible to create other views:-

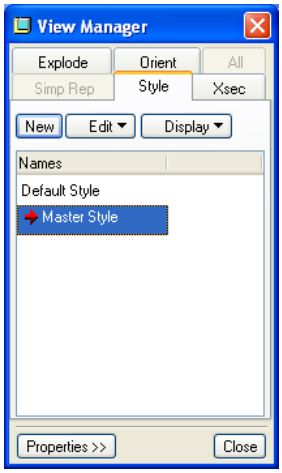
- Offset Sections (Stepped Section)
- Revolved Sections
- Breakouts
- Broken Views
- Auxiliary Views

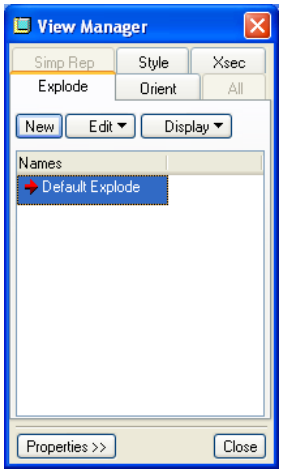
To cover these other aspects of detail drawing visit the COACH Training site at:-

http://schools.ptculms.com/cadtrain/coachlms/client/login?eventType=logout

To create a customised exploded view

There a several ways to create a new exploded view but all are associated with the **View Manager**. This multi functional tool allows you to create Xsec (Cross-sectional), Styles, Orientations and simplified representations (SimpRep).

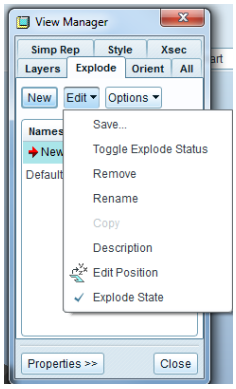


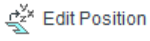


To create a new exploded view you are going to do three things.

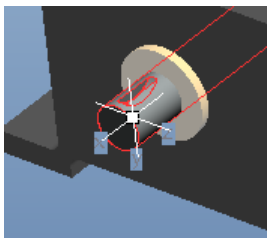
- Select whether to move one part at a time or multiple parts
- Select the direction in which they are to move
- Drag the parts to their required position

1. To start open the **View Manager** and **Left-Click** on the **Explode** tab.
2. **Right-Click** on the **Default Explode** and select **Copy**, accept the name given (Exp001) or change it to something you want. **Left-Click** OK.
3. **Right-Click** on the new name and select **Set Active**, the red arrow will appear on the left and the model will explode in the same way as the default. You are now going to edit this to create a custom exploded view



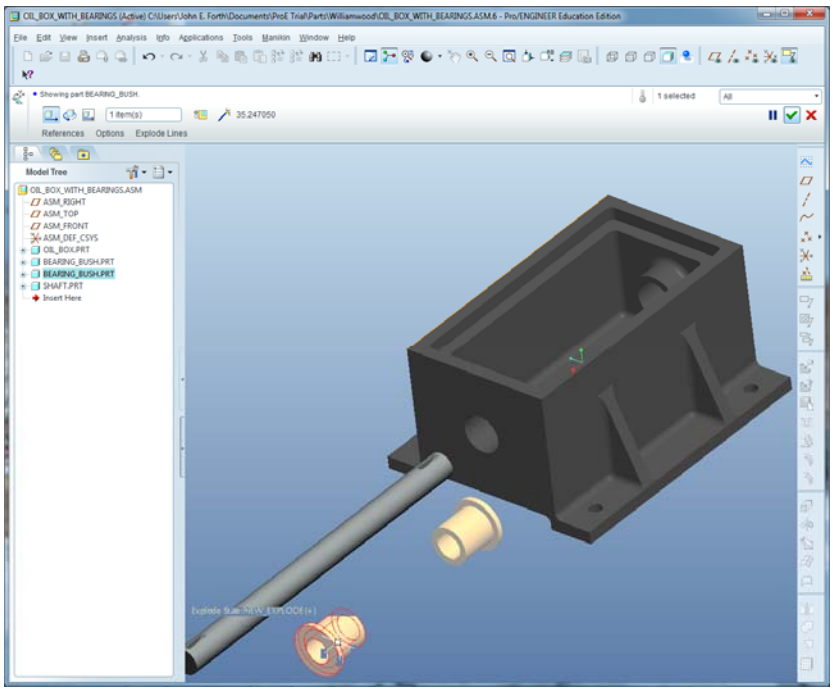
4. Left-Click the down arrow to the right of Edit. (as opposite)
5. Left-Click Edit Position. 

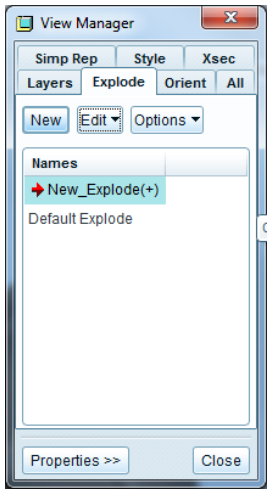
6. Now Left-Click in the graphics window on the part you want to move. A coordinate axis appears. you float the mouse over it the three coordinates highlight in turn. Select the one required for the drag direction .



7. Left-Click-Drag to move the part in the direction required.

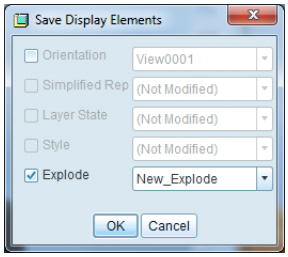
8. Position all of the parts as required then Left-Click the Green Tick in the dashboard





Take your time creating the exploded view as it will be used in the detail drawing. Tip: Create several different Exploded Views to cove all the options; don't try to do too much with each one.

9. You will notice the + sign next to name of the view in the view manager. This is an indication that to continue you need to save the view. **Left-Click** Edit and select **Save** from the drop down menu.

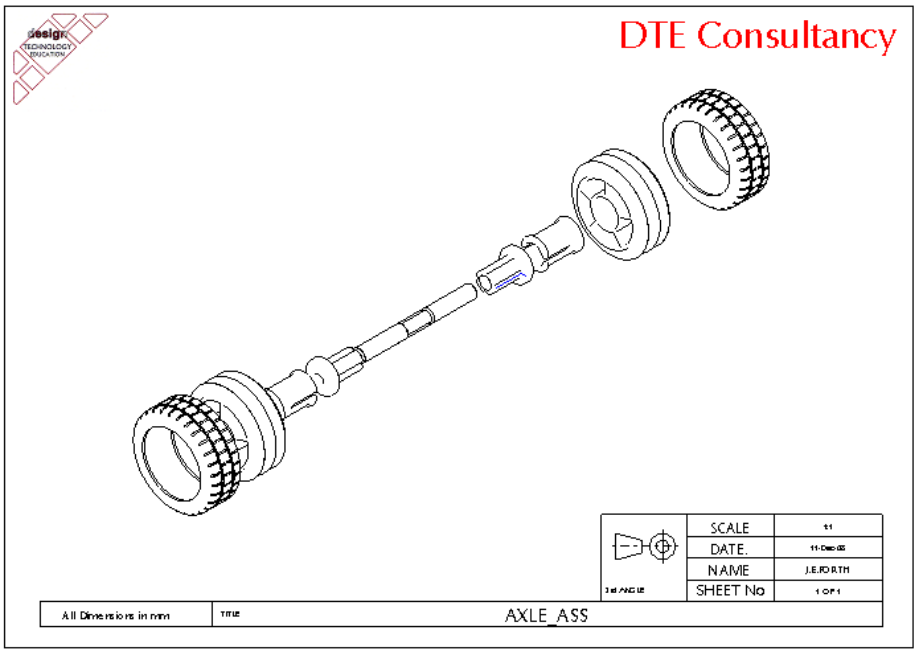


10. Make sure the name is correct and Left-Click OK.

The exploded view is now saved and can be called at any time in the graphics window and in any drawing created from the assembly.

To call the view open the View Manager and Double Left-Click the view you want to select. Go to View<Explode and select Explode View from the slide out menu. Reverse the procedure to un-explode the view.

You can select a number of ways to edit the position to explode the parts, experiment to find the one that suits your requirement.

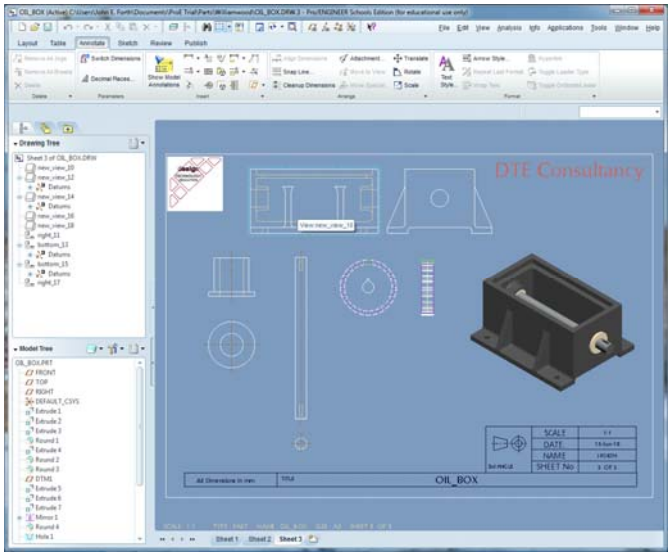


Example of a custom exploded view.

More with Annotation.

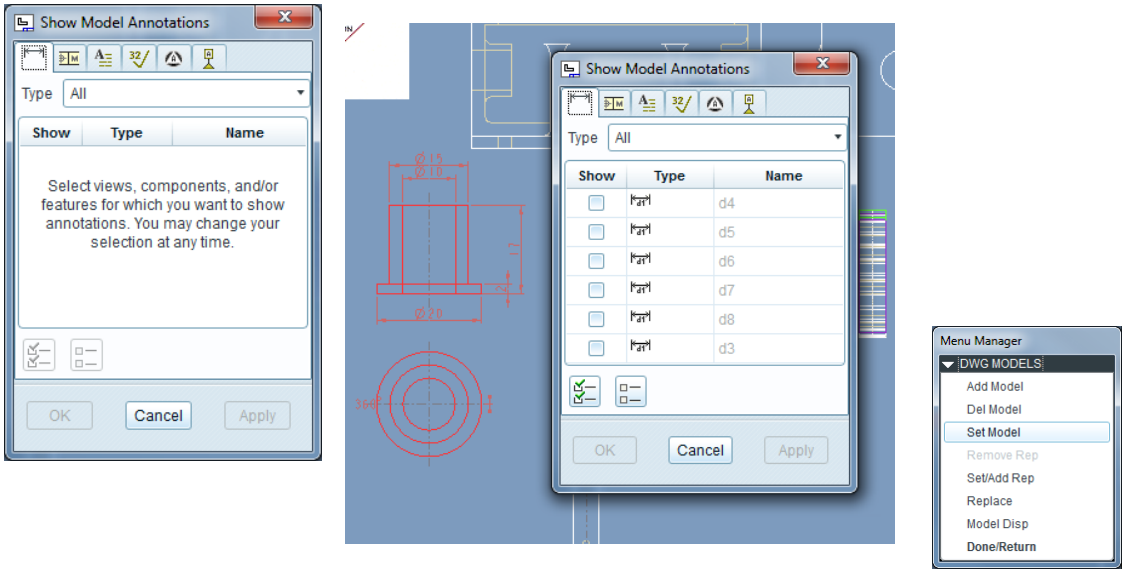
The Annotation Tab allows a wide range of actions with selected views. Especially the Show Model Annotations Icon .

On a sheet like that left you must first Set the Model in the Layout tab. As can be seen here the OIL_BOX.prt is the selected model as its model tree is visible.



To change to another model

1. Left-Click File<Drawing Models.
2. Select Set Model and from the list presented choose the model you wish to annotate.
3. To annotate that model Left-Click Follow the instructions in the box that appears. Left-Click the part name in the Model Tree.

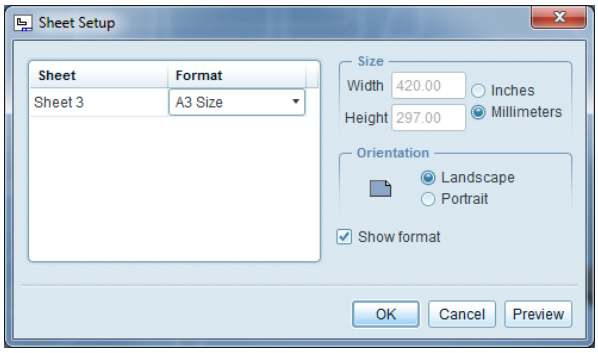


The first tab is dimensions and they appear with the views associated with this part.

4. Place a tick in the box next to the dimension required, it will not be fixed until you exit the process. Selected dimensions change colour.
5. Left-Click any of the other tabs to add those elements if they are present in the model.
6. Left-Click OK to complete the process.

To Change the Format Sheet

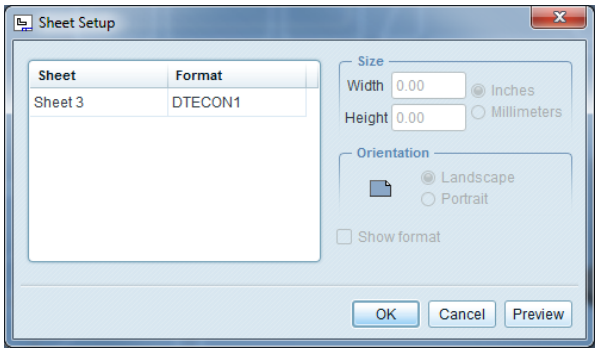
If you have created a drawing on a generic format and want to change it onto a new format it is quite simple.



1. **Left-Click File<Sheet Setup**

The Sheet setup dialogue opens

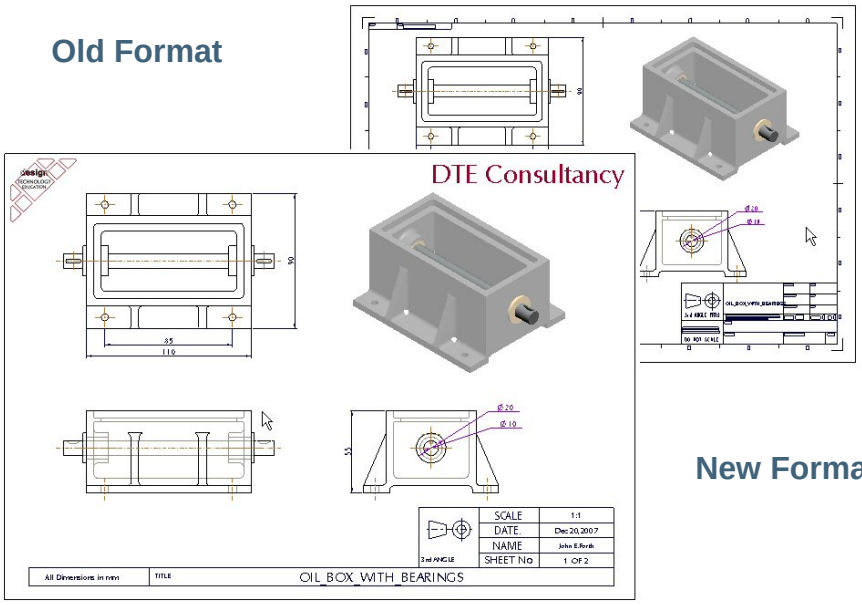
2. **Left-Click** on the down arrow to the right of format and either select the new one from the list supplied or select browse and find the one you want. **Left-Click** OK



3. Follow the instructions supplied at the top of the screen, if they appear. These will usually be to keep or remove tables on the existing format. A blank **<CR>** will retain them.

4. **Left-Click** the green tick in each case. The new format will appear with all your views in place. There is no need to alter anything unless you want to or, in the case of a sheet size change, larger or smaller, then some view movement will be required.

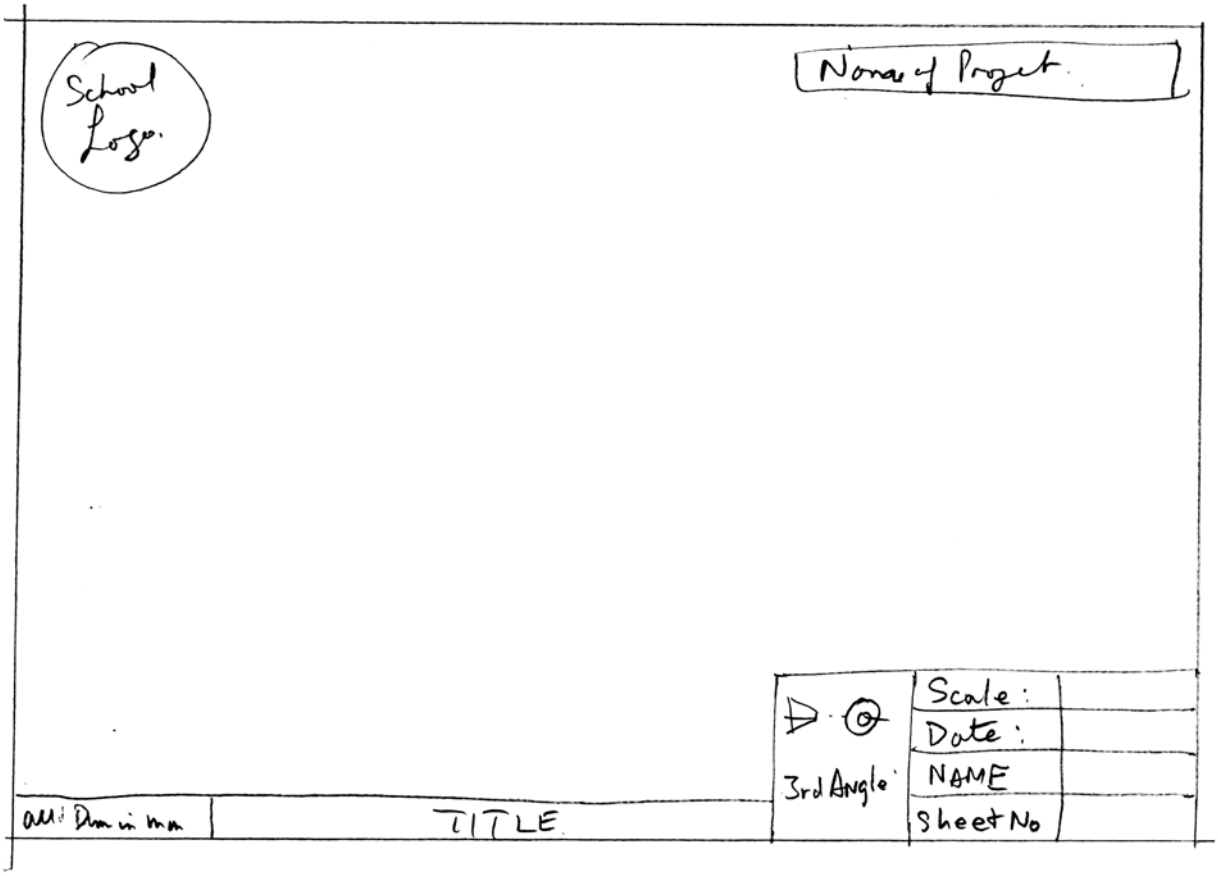
Old Format



New Format

To Create your own Format

The first thing you need to do if you want your own customised Format is to decide what it is going to look like. The best way the do that is to draw one out roughly on the relevant sized sheet of paper. As an example we are going to create an A4 sized Landscape format. Here is the rough sketch.

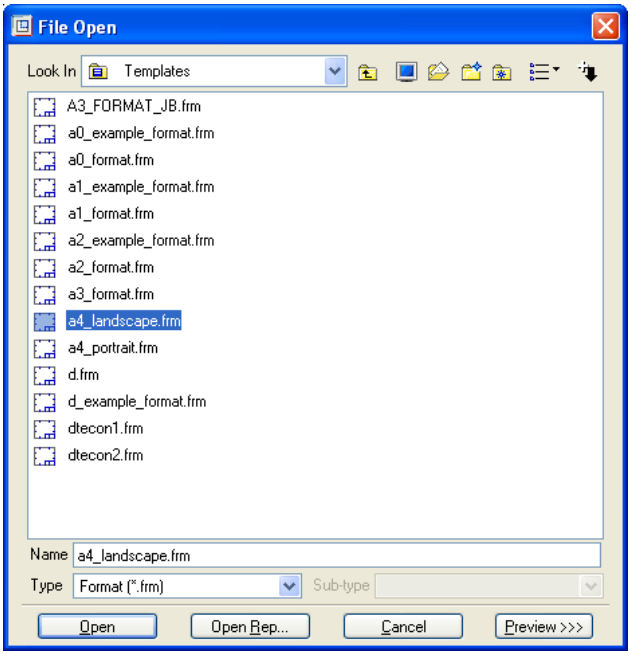


This is the guide to help you. You now have to determine how each element will be made. You can start with an existing format and make alterations or you can start with a 'clean slate'. In this case we are going to use a PTC format and modify it.

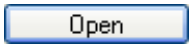
Let's look at the elements we intend to add.

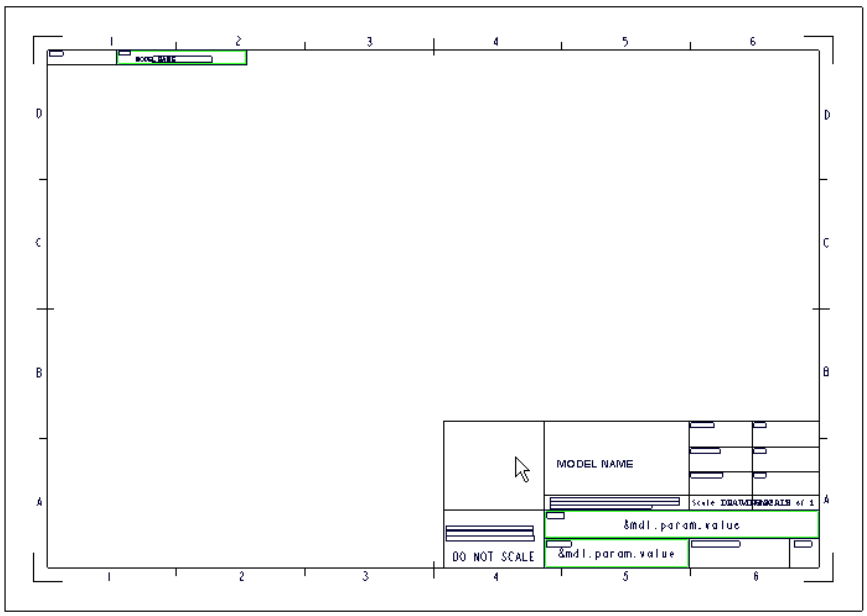
- A school or class logo; this needs to be a bitmap file and you need to know where it is so that you can retrieve it.
- A border; this can be drawn using the sketching tools or modified from the existing format.
- A title or Project name; this will be text and can be added as a note and formatted with the tools available.
- A Table; this is a block of cells that contain written or symbolic information. A Table in **Creo^{tm} Parametric**is a powerful entity, not just a series of boxes. The boxes can be 'programmed' so that specific information can appear automatically or the user can be prompted to input specific information. (see Appendix 2)
- A Symbol; this is the universal symbol for 3rd Angle projection and exists as a **Creo^{tm} Parametric** symbol.

Let us start with the basic format that we are going to change.



1. **Left-Click File<Open**
2. Navigate to the Templates folder.
3. Click on the down arrow at the top and **Left-Click Templates**
4. Set file type to **Formats**
5. Select the one you want and **Left-Click**





Let us take a good look at this Format to see what we can keep and what we need to change or delete to create our own format.

First of all notice the parts outlined in green. These are **Tables** and contain text that is interpreted by the program and will return specific parameters to the drawing. All of these parameters are in the Help files and can be found by going to Detailed Drawings in the **Help Centre** and typing **Parameters** in the **Index** Tab. (see Appendix 2) We are going to use some of these later. These parameters MUST be in a Table to be interpreted.

The other aspects such as Border, other lines, fold tabs and reference characters are all able to be deleted or left as you see fit. Save a copy of this Format and then retrieve it

to become the new format.

We will start to make the format as we want by removing the redundant items.

6. Select the items to delete by **LEFT-CLICK**. Hold down the **Ctrl** key to multiple select.
7. Hit the **Delete** Key

There will be elements you select which will not delete in this way. These are Tables and must be treated slightly differently.

To delete a table

1. Select the cell first with a **Left-Click**
2. **Right-Click** and Select '**Delete Contents'**.

Some text will remain; this is just a note to tell you what is in the cell. You can remove it later.

3. Select the Table(s) with a **Left-Click** using **Ctrl** if necessary.
4. **Left-Click Table<Select<Table**
5. **Right-Click** Select **Delete**

The table will disappear.

You can now select any text or other odd lines you do not want and delete them

If there are tables you want to keep and use they can be saved and brought back when required. To do this:-

6. Select the Table
7. **Left-Click Table<Save Table<As Table File**
8. Give it an easily remembered name such as 'title_block'

Once saved, they can be deleted as outlined before.

To retrieve the Table.

1. **Left-Click** Table<Insert<Table from File
2. Select the table you require and position using options from the GET POINTS Menu

Now we can start building up the Format. These are the steps.

1. Draw any lines you require, you can use the drawing tools to the right of the Graphics screen in much the same way as in the Sketcher.

   A useful tip is to set a draft grid.

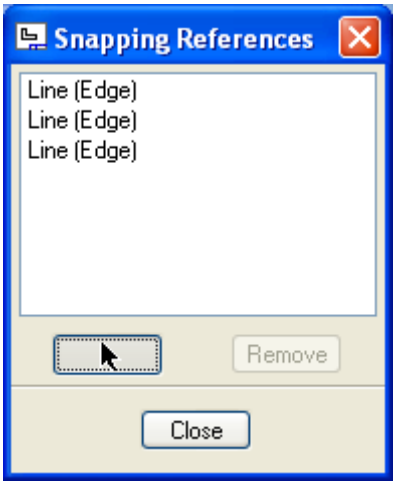
   a. **View** -> **Draft Grid** -> **Grid Params**.

   b. Set **H/V & grid snap Sketch** -> **Sketcher Preferences**.

2. Create any text as notes and place them
3. Create any Tables and populate them with parameters
4. Place any Logo's

To start with we are going to create a box at the bottom of the sheet using the line drawing tool.

5. First Select the Line Tool
6. A snapping reference box opens.
   Left-Click the arrow button

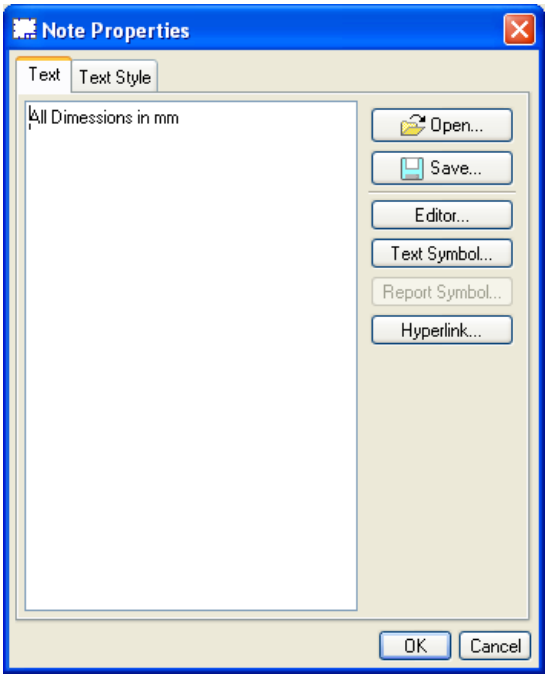


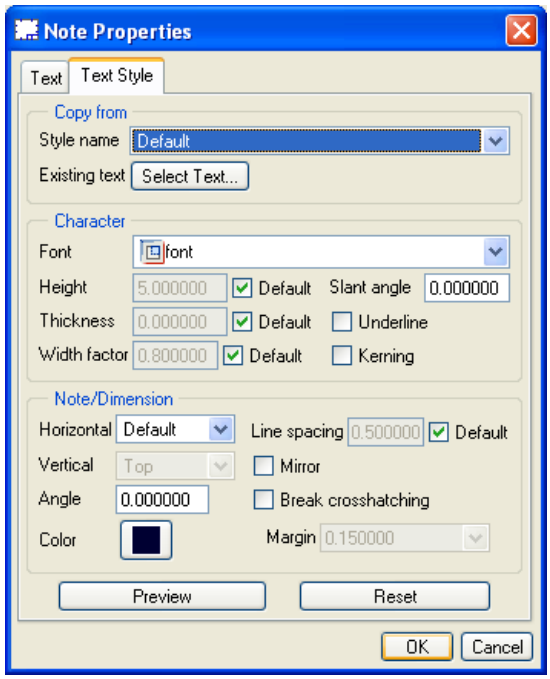
Select the two vertical lines of the border as Snapping References.

7. Draw a horizontal line between the two. **Middle-Click** to finish.
8. Drag the Line into the correct vertical position.
9. Repeat the actions to create a vertical line to enclose the units note.

The sheet now looks like this.

10. **Left-Click** Insert<Note
11. A menu manager opens.
12. **Left-Click Make Note** to accept the default or make choices as required.
13. **Left-Click** where you want the note. Approximate position is all that is required since it can be **Left-Click-Drag** to exact position later.
14. In the box at the bottom of the screen type in the note. If it is only one line you will need to **Left-Click** the green tick on a blank line. In this example type '**All Dimensions in mm**'
15. **Left-Click** Done in the Menu Manager
16. The note will now appear in the default size text set in the drawing set up file, to change this (see Appendix 1).
17. **Double-Left-Click** on the Text and the **Note Properties** box opens. Here we can change and edit the note to be exactly what we want. Note: spelling correction needed.





1. **Left-Click** the Text style Tab, here we can set the size of Text, font used and colour of the type.
2. Make changes and use the Preview button to see what you get.
3. **Left-Click OK** when all is done.

Notice that there is no formatting of the text in a plain note. If you want the text to have a box round it prefix the text with **@[**.

Continue to add text as required.

To add a Table

1. **Left-Click Table<Insert<Table** (or **Table from file** to retrieve a saved version, useful if you are creating a range of formats with similar tables).

A Menu Manager opens so that you can set out how you like you table to look. In this case we are going to create an ascending table Leftward, By Length. We are going to lock it to the right border and the horizontal Line we created earlier by using their Vertex, i.e. the corner. This will create a table starting at the bottom right hand corner of the sheet. Looking back at the rough sketch we need a table of 2 columns and 4 rows. We will be using drawing units to give the size; it is easier than working out how many characters you will put in the cell.

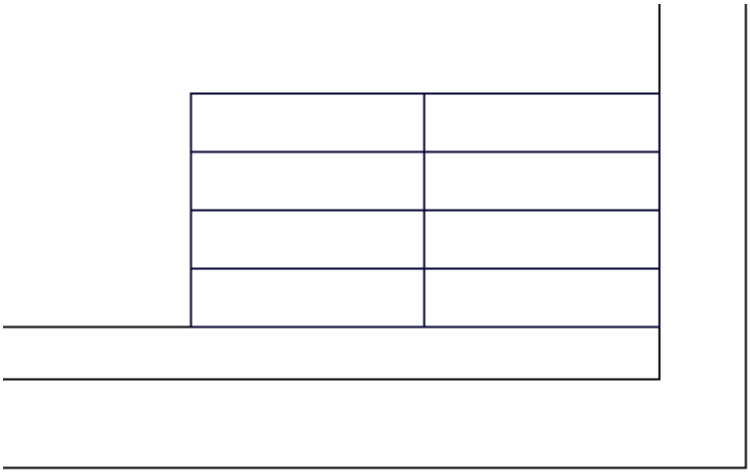
1.. Select the Vertex. You will be prompted to enter the width of the first column.

2. Type in the width in drawing units e.g. mm and either **Left-Click** the green tick or hit the Enter key.

3. The next column width is the entered and so on until you have defined all of the columns you need. In this case stop after two columns are defined.

4. To stop setting columns; **<CR> Enter** on a blank prompt.

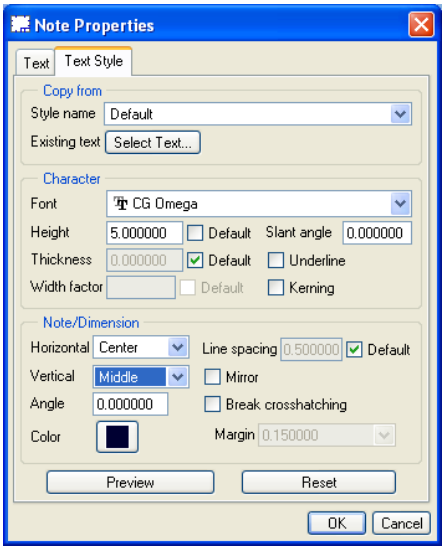
5. You will now be asked to define the row heights in the same way, finishing with a **<CR> (Enter)** after the last one. Your Table will then appear.



You can create other tables in the same way; instead of drawing a box to put the symbol in we will use a 1 X 1 table.

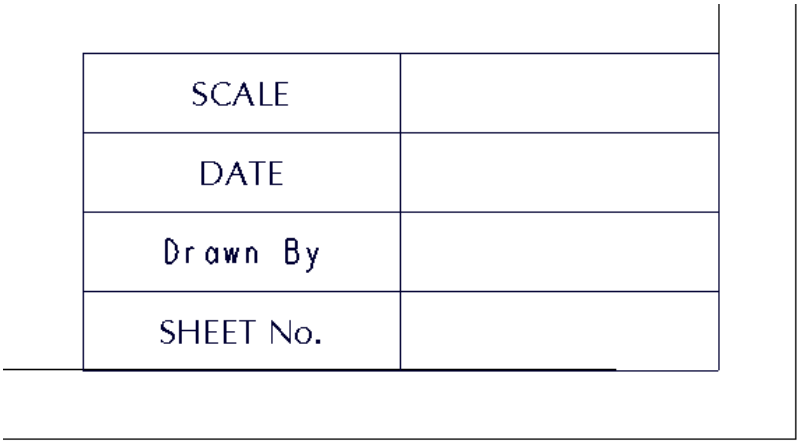
2. Add text to the left hand column using the table function.
3. **Left-Click** in the cell, the **Note Properties** box opens.
4. Enter here the text, if you want to use any special symbols open the **Text Symbols** box.
5. Open the **Text style** tab.

Here you can add font, size, colour and position in the cell, etc.,



Use the **Preview** button to see the affect of any changes before you commit.

6. Now it looks like this, you can edit any cell by selecting it and **Double-Left-Click** to return to the **Note Properties**.



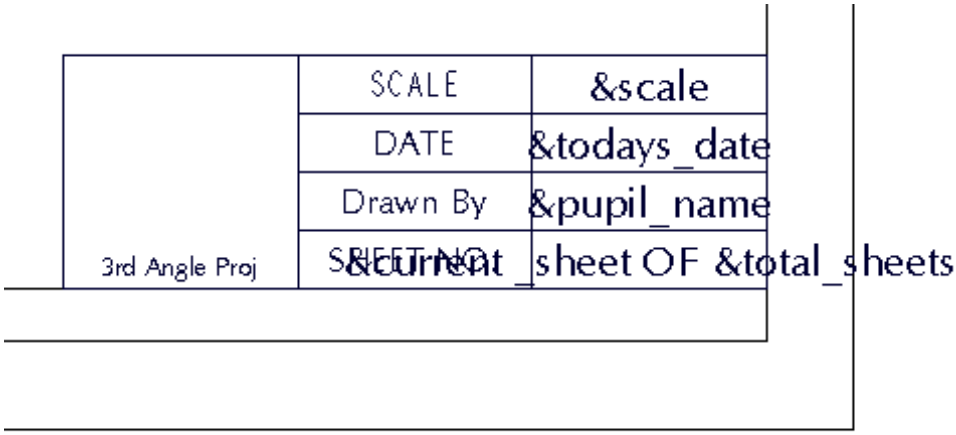
You can now add parameters to the other cells that will be interpreted by the program to enter the Date, Scale and Drawing number. These are of two types, those automatically placed by **Creo™ Parametric** and those requiring action on the part of the user. We will deal with the automatic ones first.

The ones needed in this example are for Scale which is **"&scale"**; Date which is **"&todays_date"**; Sheet number which is **"&current_sheet of &total_sheets"**. These you will note have an ampersand preceding the text and no spaces; an underscore is

used as in the case of a file name. (For other parameters see Appendix 2). These are placed and formatted in exactly the same way as the earlier notes. Enter everything between the quotes. See Appendix 2 for other parameters you may need.

You can enter a user parameter in the same way but this will elicit a user response. In this case the Pupils name in the '**Drawn By'** cell. We are going to use the parameter **"&pupils_name**". Whatever word or words you use here will be the user prompt when the format is employed.

Your Table will now look like this:-

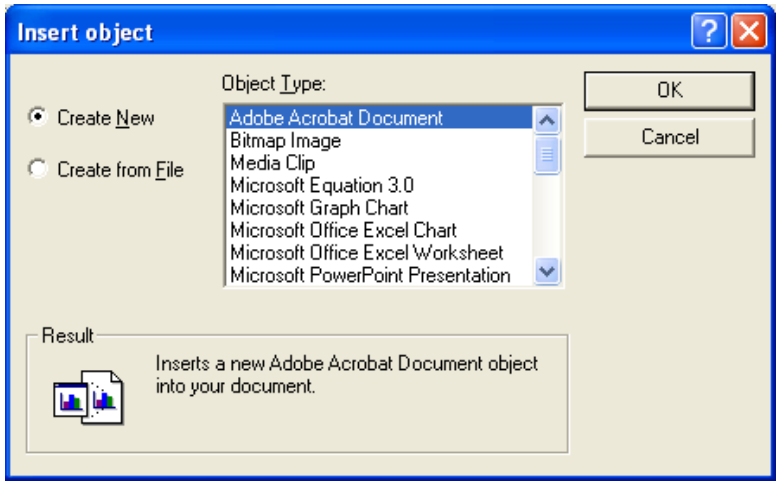


(Note: On the format they will appear as typed; when the format is used the relevant information will appear. Don't worry if they overlap the cell, as the value returned will not.)

To Add a Logo

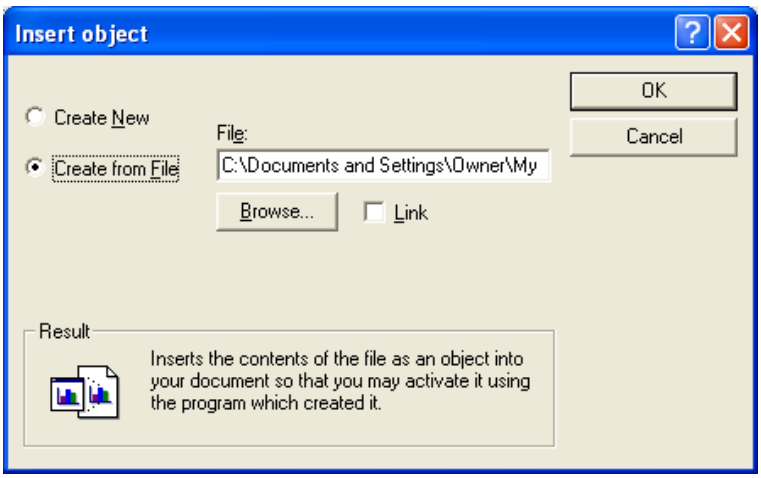
The School badge, Team or Project Logo can be added to the format or drawing, either in a cell or not. In this example the logo will go into the top left hand corner of the sheet. It must be a Windows bitmap file (*.bmp)

1. **Left-Click Insert<Object**
2. It is better to have it already created so select **Create from File**.



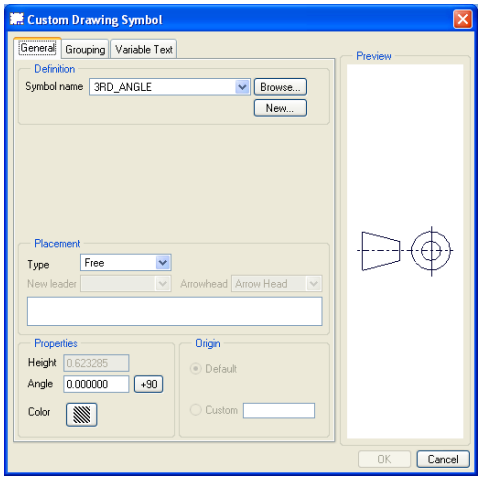
3. Browse to the file and **Left-Click** Ok

4. **Left-Click** on the Graphics Window to place the bitmap image. Size the image using the drag corners and reposition as required.



To Place a Symbol

All that remains now is to place the symbol that shows it is a 3rd Angle Drawing.



1. **Left-Click Insert<Drawing Symbol<Custom**

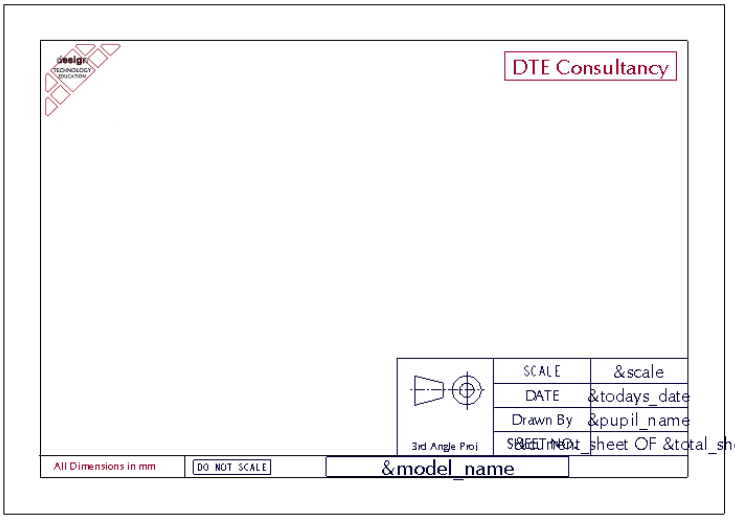
2. Browse to the Folder containing the symbol and select it. This box appears.

A ghost image of the symbol now appears at the mouse cursor.

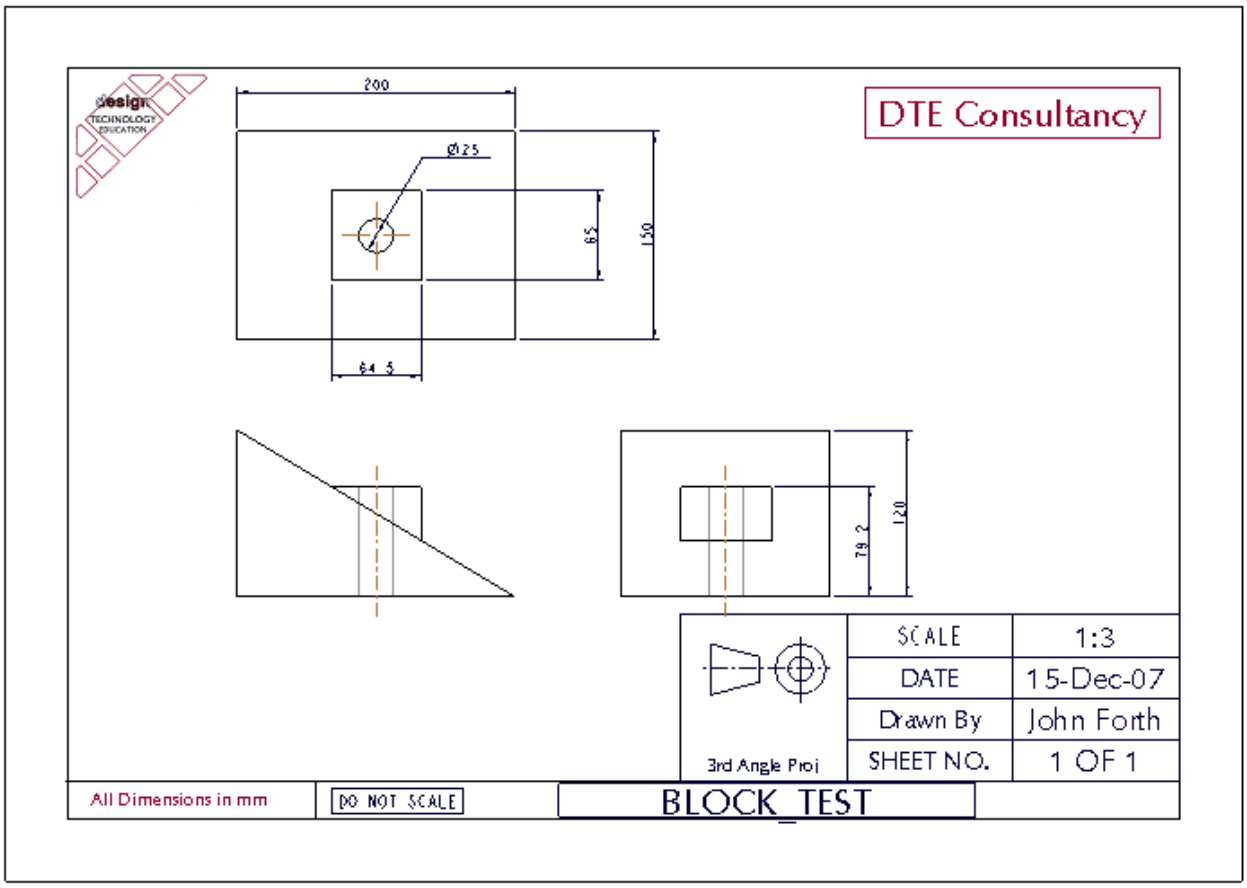
3. Position the symbol and **Left-Click** to place.

4. **Left-Click** Ok

The Format is now complete. Save the file in the Templates Folder or a Templates Folder in your user area. You can of course retrieve it at any anytime and modify it.



This format can now be used to create your own unique set of drawings. Here is what it looks like when a model is shown.



These views have been manually placed, now you can use this format to create a Template onto which the views will be placed automatically.

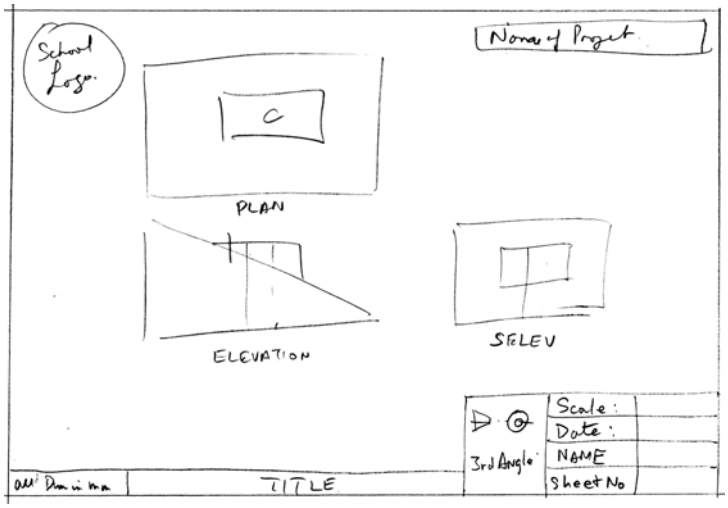
To Create a Template.

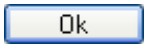
The template is the file that sets the desired views and their display state; create snap lines; and shows model dimensions based on the way the template is created.

As an example we will create a Template based on an A4 format that will show three views in 3rd Angle projection, two of which will have hidden detail showing and none will show tangent edges. You can draw a rough sketch to determine how you want the views to look.

1. **Left-Click New Drawing**
2. Give the template a name e.g. DTE_A4_TEMPLATE.

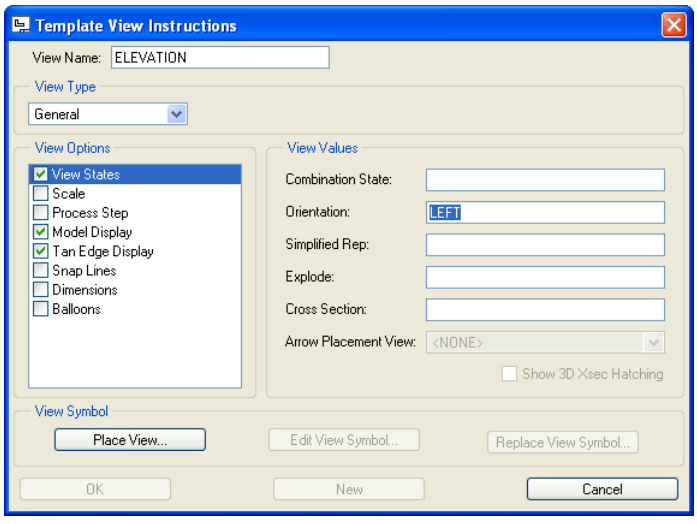
The name should tell as much as possible about the template.



3. Select the radio button **Empty with Format**
4. Browse to find the format you want to use. In this case we will use the one created earlier.
5. **Left-Click** 

6. **Left-Click Applications< Template**

We now have to place the views on the sheet. In this case the first one is the **ELEVATION** which will be a **General view**; Orientated **LEFT** (see View Manager); **Hidden**; **No Tangent Edges**.



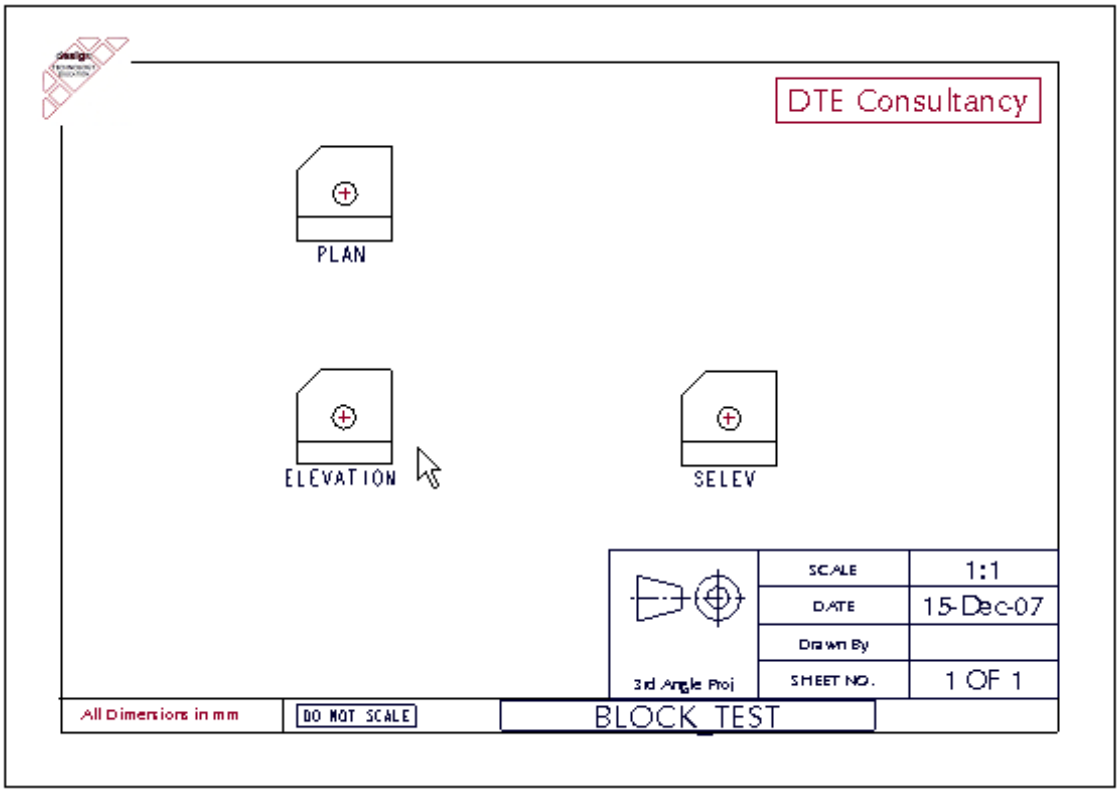
7. **Left-Click Insert<Template View** The Template View Instructions box appears. In this we can first of all set the View Name --**ELEVATION**

Now set the Orientation –**LEFT** (Note there are many options here, what you use depends on the template you are creating. The Views are those available in the View Manager)

Notice, under 'View Options' there are some boxes ticked and others not. View states MUST be ticked the others are optional depending on your template. In this simple

example we will only use those options checked.

8. **Left-Click** '**Model Display'** select "**Hidden**" to show hidden detail
9. **Left-Click** '**Tan Edge Display'** select "**No Disp Tan**" to not show tangent edges
10. **Left-Click** 'Place View', **Left-Click** on the Graphics Window to place the view. It can be reselected to reposition it if required.
11. **Left-Click** '**New'** to set up the next view; this will be a projection from the ELEVATION. **Left-Click** the down arrow to the right of the View Type box and select **Projection**. Ensure that ELEVATION is shown in the Parent View Name box. Set Model Display and Tan Edge Display as before and name the view PLAN. Place the view.
12. **Left-Click** 'New' to set up the last view, a Side Elevation Projected from the ELEVATION. **No Hidden** this time.
13. You can reposition the views if you wish and by **Double-Left-Click** on the symbol you can edit the view display.



The views and the view names appear in this view but will be replaced by the views of whatever model you use when creating your drawings.

The last thing you need to do is to re-allocate the two parameters '&scale' and '&model_name' .

14. **Left-Click** to select the cell **Double-Left-Click** and change what is there to the **Creo^{tm} Parametric** parameters namely **'&scale'** and **'&model_name'**.

They will appear in the template as "**DRAWING SCALE**" and "**MODEL NAME**" this will appear as the correct scale and model name when the Template is used.

15. Setup the drawing option file as described in Appendix 1.

Before saving the Template file you will need to make sure that no models are associated with it.

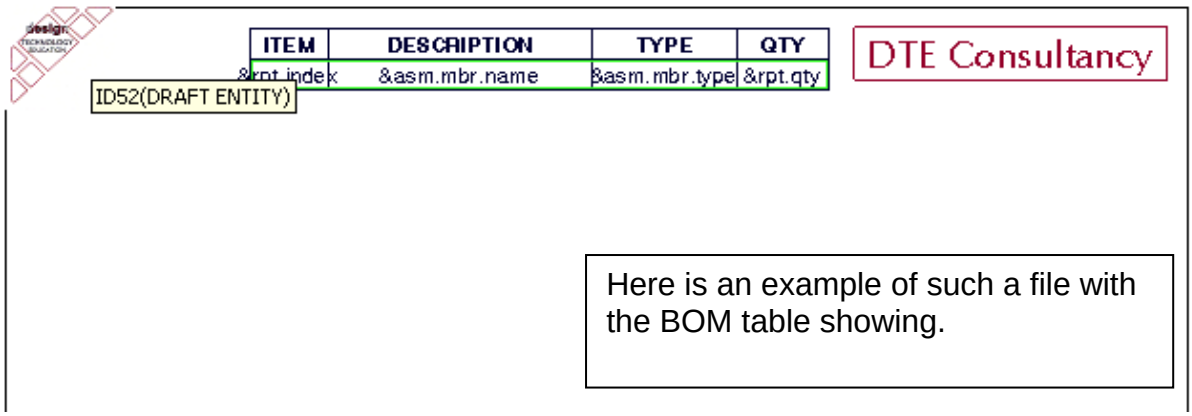
16. **Left-Click File<Properties<Drawing Models**
17. Refer to the dashboard area. If a model is associated it will be noted with the option to retain or delete. **Left-Click** "Yes" to delete
18. Save the template file.

To Add a Bill of Materials (BOM) Table.

A Bill of Materials Table is a special case of the use of a Table in drawings. The table is created to show each of the components within the assembly plus other information that may be required. Each cell in the table has a parameter and each cell can be made to duplicate itself as many times as is required to cover any size assembly. Cells can also be set to create Numbered Balloons attached to the parts.

The easiest way to add a BOM table to your format is to copy one from an existing PTC format found in the Templates folder. This one is a simple example but adequate; if you want to create a more complex one, see the Help Centre files.

1. Set **Templates** Folder as the **Working directory**
2. Open the format file with the BOM table



3. **Left-Click** to select the Table
4. Left-Click **Table<Save Table<as Table File**
5. Give it a name e.g. bom.tbl

This will save the table in the Templates Folder

6. Close the file and **File**> **Erase<Not Displayed** to clear the memory
7. Open your format file.

8. Left-Click **Table**<**Insert**<**Table from File**
9. Place the Table using a **Left-Click-Drag**
10. **Left-Click** to finalise.
11. **Save** the format file

To set the table to create balloons

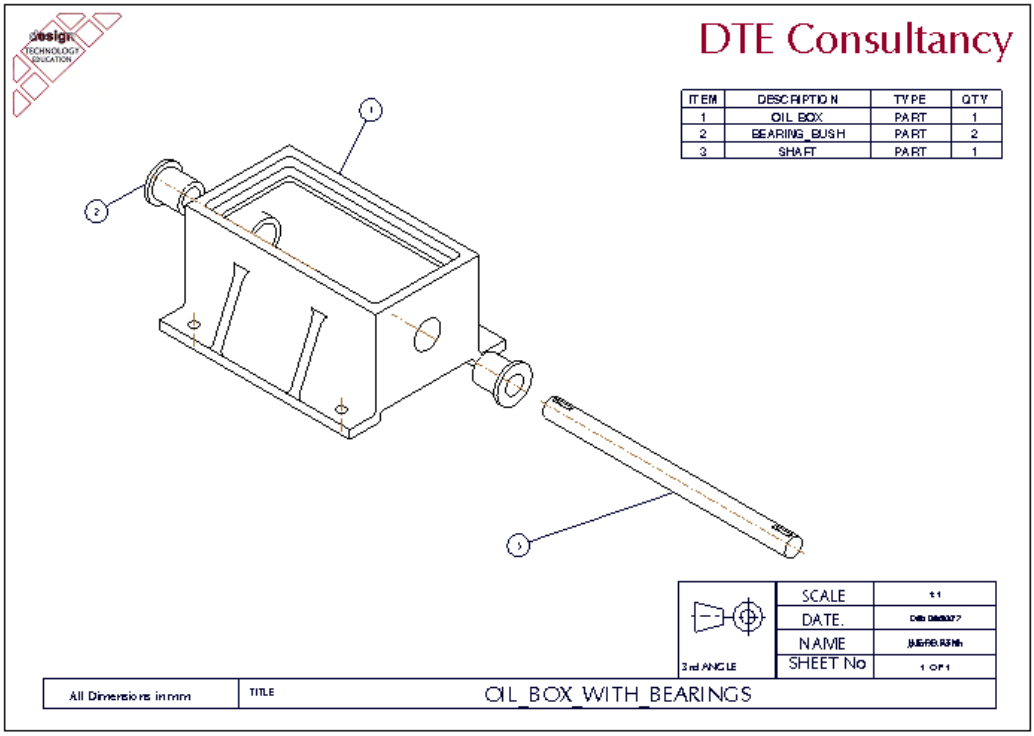
1. With the format open
2. **Left-Click Table<BOM Balloons**, BOM BALLOONS Menu Manager opens.
3. **Left-Click** the Cell '**ITEMS'** This is where the part no. is stored
4. **Left-Click Simple** to provide a balloon with just the number.
5. Left-Click **Done**
6. Save the format

To Show Balloons in an assembly View

1. **Left-Click Table**<**BOM Balloons**
2. **Left-Click Create Balloons** BOM VIEW Menu appears
3. **Left-Click** Show All
4. Left-Click **Done**
5. **Drag-Left-Click** Balloons into place

You will notice that the table only has one row and is populated with parameters that start **'&asm**_' which collects the data required from the part file. The table, when used, will display all of the components that comprise the assembly. The table is designed to extend to the size required.

Format with BOM table used

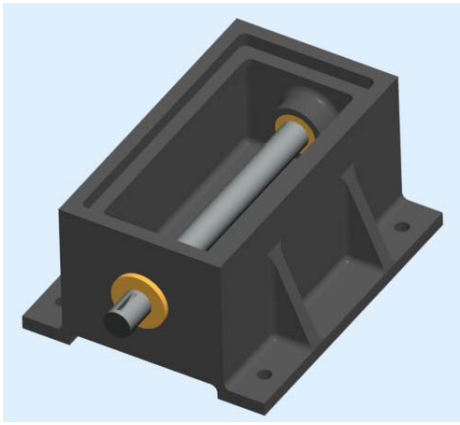


If you wish to place a BOM table into an existing drawing or on format without a bom table in place then follow the procedure in the next section.

Creating a Bill of Materials for a blank format.

Creating a BOM in a drawing is a three step process starting with an assembly open in **Pro**|ENGINEER.

- In the assembly create the bill of materials
- In the drawing of the assembly insert a BOM table
- Create BOM balloons for components



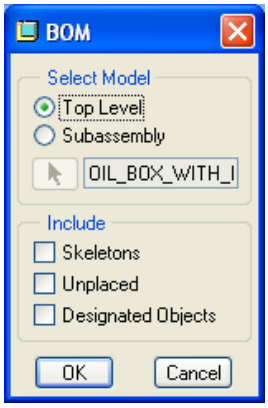
Creating the BOM for an assembly

1. Your assembly should be open in **Pro**|ENGINEER.

2. In standard mode…the BOM will display in the browser.

3. In the main toolbar across the top of the screen open the **Info** menu and select **Bill of Materials**

4. The BOM dialog opens

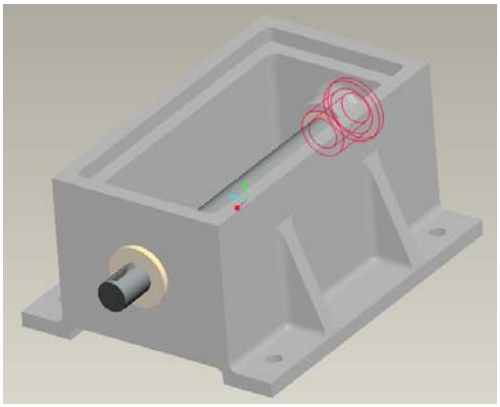


5. Click **OK**

6. The Browser in **Pro**|ENGINEER will open displaying the Bill of Materials for the assembly.

The BOM lists all the components, the quantity where there are multiples.

The columns give access to information about components and by clicking on a part it highlights in the graphics window.



Bom Report : OIL_BOX_WITH_BEARINGS

Assembly OIL_BOX_WITH_BEARINGS contains:

Quantity	Type	Name	Actions
1	Part	OIL_BOX	
2	Part	BEARING_BUSH	
1	Part	SHAFT	

Summary of parts for assembly OIL_BOX_WITH_BEARINGS:

Quantity	Type	Name	Actions
1	Part	OIL_BOX	
2	Part	BEARING_BUSH	
1	Part	SHAFT	

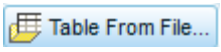
BOM tables

BOM tables are created from assemblies in HTML format and before they can be inserted into drawings you will need a template table file. You can download one called bom.tbl from the PTC Education Windchill Project Link site.

https://pds.ptc.com/Windchill/netmarkets/jsp/project/view.jsp?oid=project%7Ewt.projmgmt.admin.Project2%3A183631245&u8=1

Note: It is possible to create your own BOM table template with the necessary 'intelligence' to pull information from the models. (see next section)

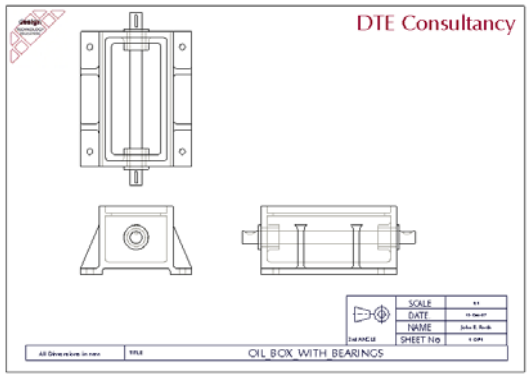
Inserting a BOM table

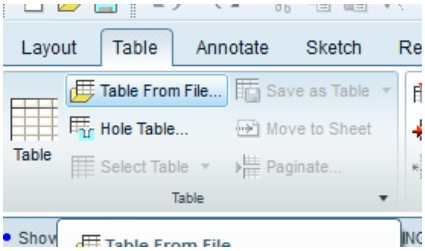
1. With your drawing open on screen,
2. In the Table Tab select **Table from file**. 

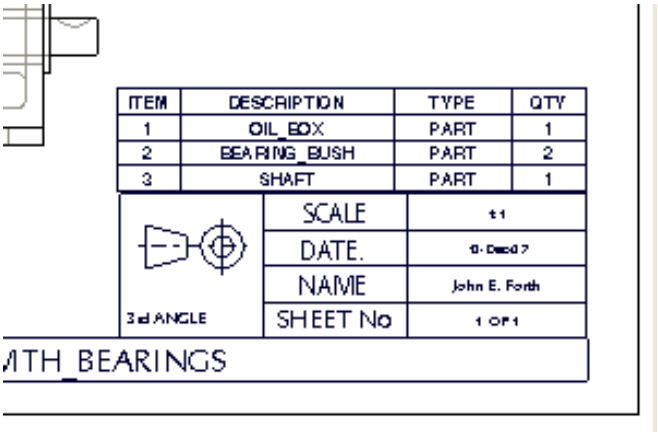
3. Browse for the **bom.tbl** and place it in the drawing somewhere.

You will see that it already has rows for each item.

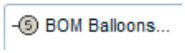
4. Drag the table into position on the drawing.



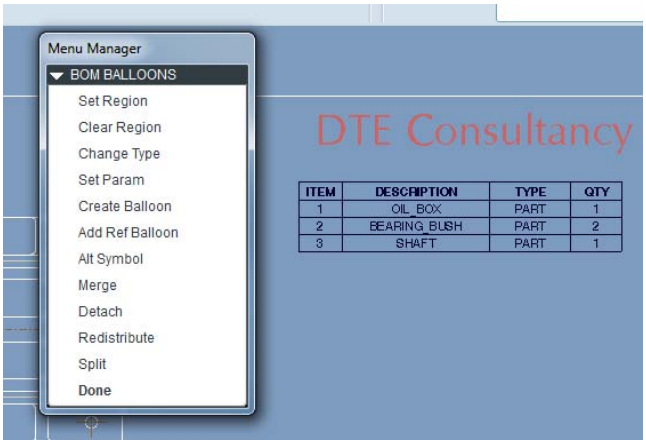




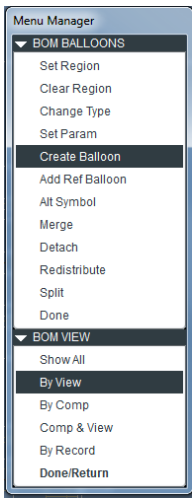
Adding BOM balloons

1. With the drawing open in **Pro**|ENGINEER.
2. In the graphics window select the view you want to display BOM balloons.
3. In the Table Tab Left –Click to select **BOM Balloons** 

4. Select **Set Region** and select the whole table.

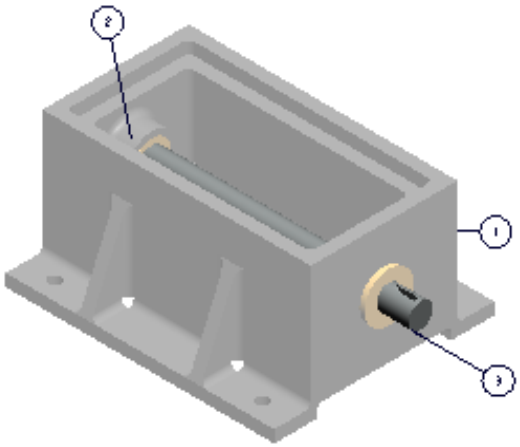


5. Select Create Balloon. By view is the default. Left-Click to select the view you want the balloons to attach. In this case the Isometric



The Isometric view will now have BOM balloons with leader arrows showing each of the components.

6. Click **Done/Return** to close the BOM Menu.

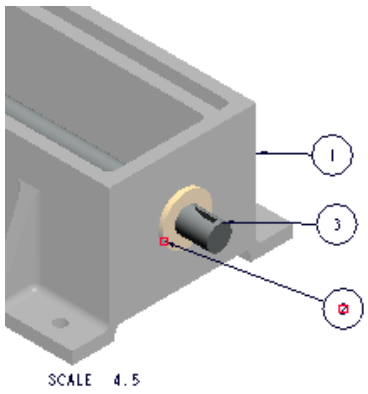


You have placed a table and all the balloons.

The balloons can be selected by **Left-Click** and dragged to reposition.

To ensure the arrow points correctly:-

1. **Left-Click** to select the arrow leader.
2. **Left-Click-Drag** the arrow head to position it on the part.



Note: You can only position the balloon on the relevant part.

To create a new Repeat Regions Table

There are two reasons to create a table that can pick up information from the model or assembly file. The first we have covered already which is the Bill of Materials or 'BOM' table. The second is a more complex table that can refer to instances of a part contained within a Family Table. The way this works is based on the principle of 'smart' table cells called repeat regions. These are user defined sections of a table that expand or contract to accommodate the amount of data that the associated model or assembly currently possesses.

1. The Extended Bill of Materials Table

The simple one we have shown so far is adequate for most projects however you may wish to add other columns, 'material used' for example.

Shown below is the simple BOM table taken from our format.

ITEM	DESCRIPTION	TYPE	QTY
&rpt.index	&asm.mbr.name	&asm.mbr.type	&rpt.qty

The top row is normal text and will appear in the drawings as just that. The bottom row contains symbols, entered as text but will appear in the drawing as the information that the symbol extracts from the model or assembly in the same way as explained in the format creating section.

When the table is updated in the drawing, this is what appears.

ITEM	DESCRIPTION	TYPE	QTY
1	OIL_BOX	PART	1
2	BEARING_BUSH	PART	2
3	SHAFT	PART	1

To create a repeated region table.

There are three parts to the procedure; the first is to create the blank table with the requisite number of columns and rows. The second part is to define the repeat regions of the table. The third is to populate the table with the plain text headings and the parameters that call up the data.

1. Create a table with 2 rows and X columns. (Where X is the number you require. You do not need any more than two rows since they will repeat as many times as is required.) See the section 'To add a Table'* for help on creating a table.
2. Populate the top row of cells with the headings required. *
3. To define the repeat region of the table.

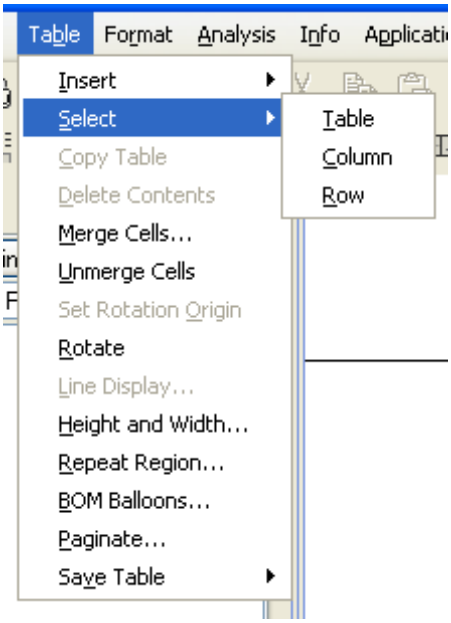
   a. **Left-Click** cell in the second row of the table.

   b. Left-Click Table

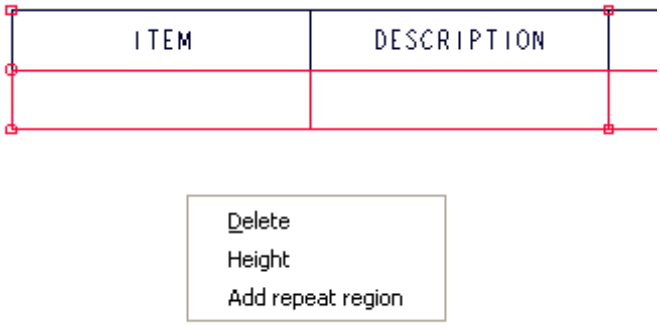
   c. **Left-Click<Select< Row** The row is now highlighted

   d. With the row still highlighted **Right-Click** and select **Add Repeat Region** from the floating menu.

   e. The row is now a set of repeat region cells

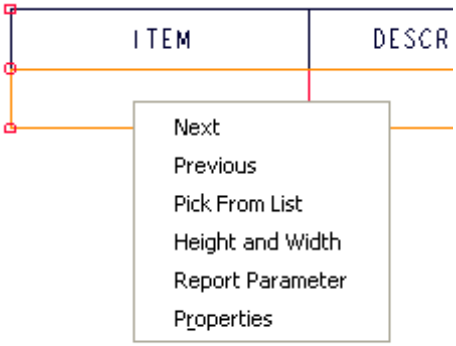


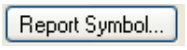
Note: A repeat region doesn't have to be a row it can be anything from a single cell to a block of several cells depending on your need.



To enter the report parameters into the repeat region

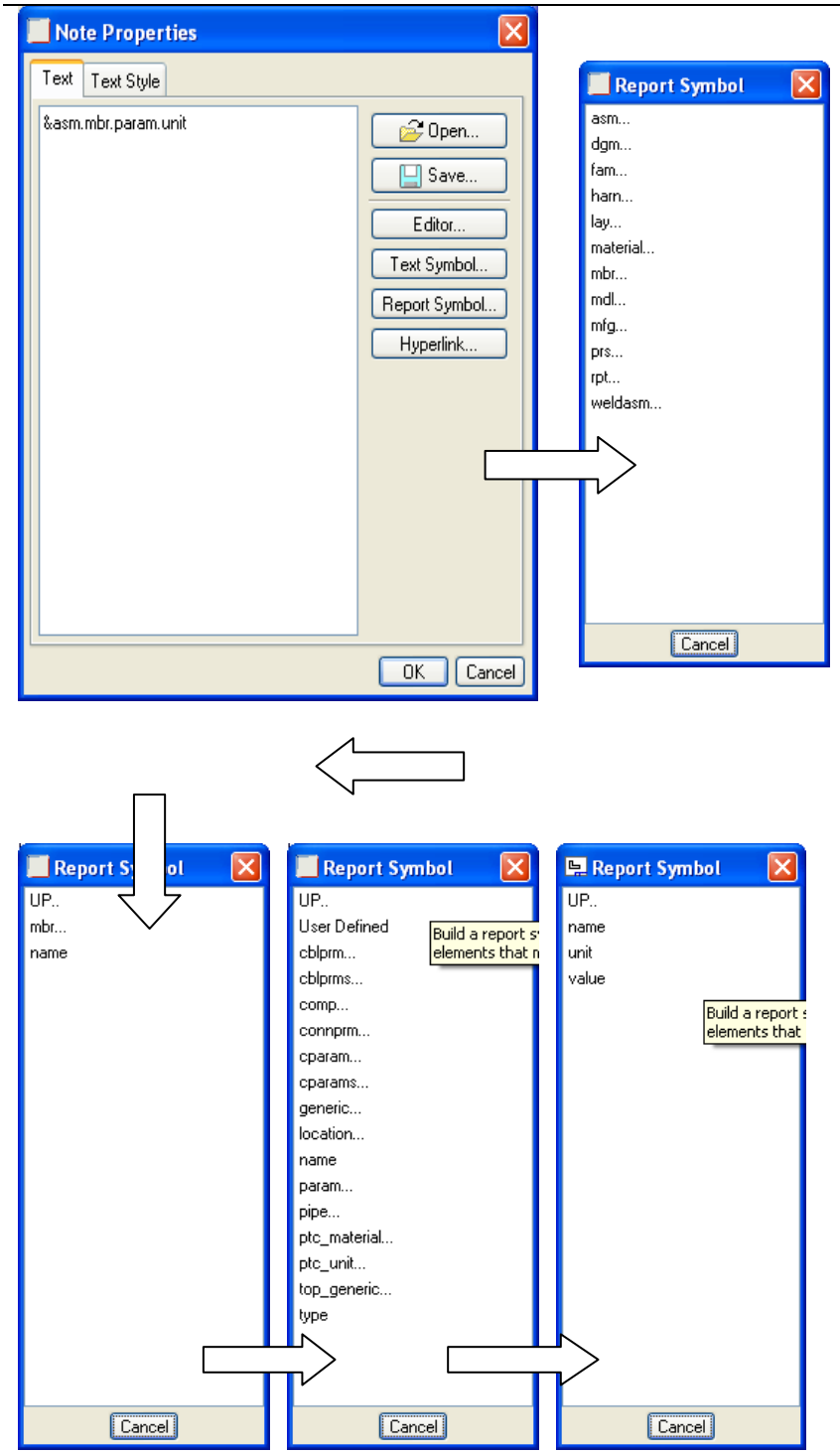
1. **Right-Click** in the target cell.
2. **Left-Click Properties** from the floating menu.



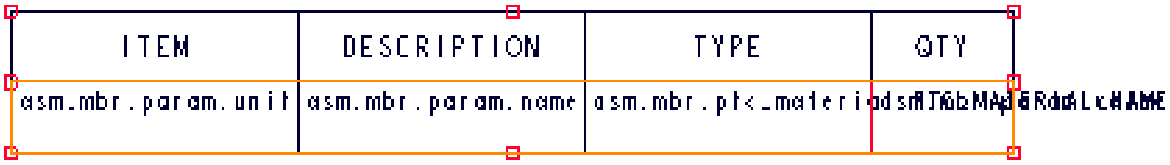
3. **Left-Click** 

4. We are now presented with a set of parameters from which to choose; depending on choice the next menu will appear. Here we have chosen '**asm**..' then '**mbr**..' then '**param**..' then '**unit**..'. The 'UP..' parameter allows us to return to the previous menu. Once all the parameters are defined the complete string will appear in the **Note Properties**. Once the whole table is complete a Double Left-Click on any parameter will bring up the symbols again. A **Right-Click** on a selected cell will bring up the **Note Properties** for editing.

5. It is possible to type in the data direct to the cell using the parameters in Appendix 2. The "&rpt" must precede any parameter used. If a **User Defined** parameter is inserted this will need to be added when the format is used in much the same way as the '&pupil_name'



4. The table will now look like this once complete.

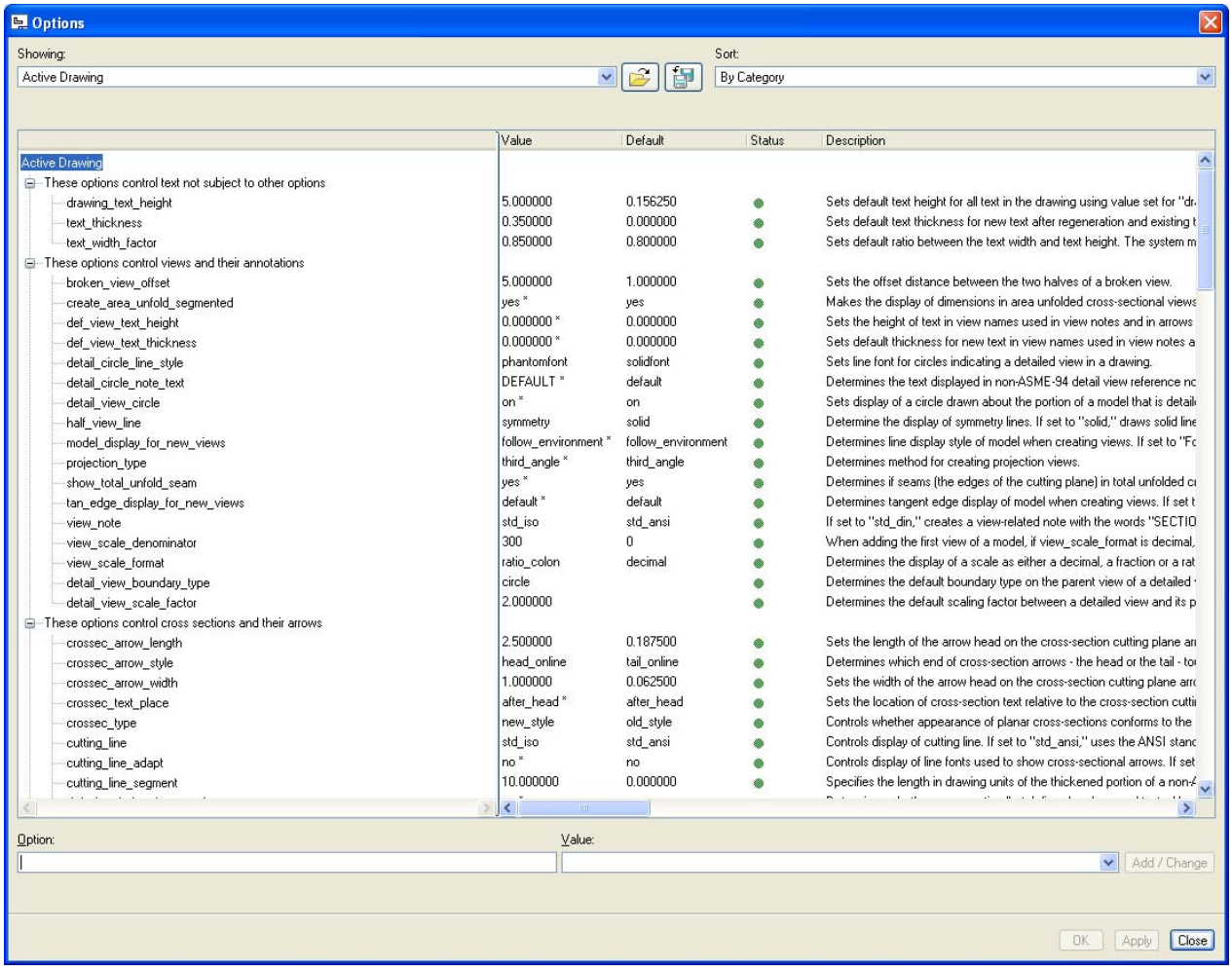


The parameters will be replaced by the data drawn from the part and assembly files, or from family tables.

Appendix 1. Setting up and editing a Drawing Options File.

The drawing options file, (*.dtl) where * may be ‘bs8888’, ‘iso’ or ‘ansi’ , exists to set the default values for many options that appear in drawing. Unless it it changed for a particular drawing the one set in the config pro file will contain all of this information. Great care should be taken if these default files are edited. However it is possible to copy this file and attach it to the active drawing or template.

1. Have the Template open on the screen
2. **Left-Click File<Properties**
3. Select **Drawing Options** from the **Menu Manager**



The file opens and can be edited. Select the option to edit and change its value in the right hand box. Information on the parameter can be read on the Right. Some options require a numerical value others have a range of options that can be selected using the drop down menu.

1. **Left-Click Add/Change<Apply**; then move on to the next parameter to alter.
2. **Left-Click** Close and **Save** the file.

The only parameters you will need to change to comply with BS8888 are as follows:-

‘drawing_units’	this is set ‘mm’ other values are affected by this.
‘drawing_text_height’	Set as a multiple of the drawing units; e.g. 5 for 5mm high text.
‘decimal_marker’	the default in COMMA change to PERIOD
‘view_scale_format’	the default is decimal, change to ratio_colon e.g. 1:2
‘view_scale_denominator’	the default is 0 change a denominator that is suitable for the drawing or a large number if many different scales are required. A good general no. is 300

You may change some of the others but I would advise caution and test as you go.

To edit the global *dtl file you can open it in a Text Editor and change the parameters manually. **CAUTION**: This could destabilise your installation

Appendix 2 System Parameters for Drawings

(Taken from the **Pro**|ENGINEER - Help Centre)

PARAMETER NAME	DEFINITION
&d#	Displays a dimension in a drawing note, where # is the dimension ID.
&ad#	Displays an associative dimension in a drawing note, where # is the dimension ID.
&rd#	Displays a reference dimension in a drawing note, where # is the dimension ID.
&p#	Displays an instance number of a pattern in a drawing note, where # is the pattern ID.
&g#	Displays a gtol in a drawing note, where # is the gtol ID.
&<param_name>	Displays a user-defined parameter value in a drawing note.
&<param_name>:att_cmp	An object parameter that indicates the parameters of the component to which a note is attached.
&<param_name>:att_edge	An object parameter that indicates the parameters of the edge to which a note is attached.
&<param_name>:att_feat	An object parameter that indicates the parameters of the feature to which a note is attached.
&<param_name>:att_mdl	An object parameter that indicates the parameters of the model to which a note is attached.
&<param_name>:att_pipe_bend	An object parameter that indicates the parameters of the pipe bend to which a note is attached.
&<param_name>:att_spool	An object parameter that indicates the parameters of the spool to which a note is attached.
&<param_name>:EID_<edge_name>	An object parameter that references edges.
&<param_name>:FID_<feat_ID>	An object parameter that includes a feature parameter in a note by ID.
&<param_name>:FID_<FEAT_NAME>	An object parameter that includes a feature

	parameter in a note by name.
&<param_name>:SID_<surface_name>	An object parameter that references surfaces.
&angular_tol_0_0	Specifies the format of angular tolerance values in a note from one to six decimal places.
&current_sheet	Displays a drawing label indicating the current sheet number.
&det_scale	Displays a drawing label indicating the scale of a detailed view. You *cannot* use this parameter in a drawing note. **Pro**|ENGINEER creates this parameter with a view and places it in notes automatically. You can modify its value, but you cannot call it out in another note.
&dtm_name	Displays datum names in a drawing note, where name is the name of a datum plane. The datum name in the note is read-only, so you cannot modify it; unlike dimensions, a datum name does not disappear from the model view if included in a note. **Pro**|ENGINEER encloses its name in a rectangle, as if it were a set datum.
&dwg_name	Displays a drawing label indicating the name of the drawing.
&format	Displays a drawing label indicating the format size (for example, A1, A0, A, B, and so forth).
&linear_tol_0_0	Specifies the format of dimensional tolerance values in a note from one to six decimal places.
&model_name	Displays a drawing label indicating the name of the model used for the drawing.
&parameter:d	Adds drawing parameters to a drawing note, where *parameter* is the parameter name and *:d* refers to the drawing. .
&pdmdb	Displays the database of origin of the model.
&pdmrev	Displays the model revision.
&pdmrev:d	Displays the revision number of the model (where *:d*

	refers to the drawing).
&pdmrl	Displays the release level of the model.
&scale	Displays a drawing label indicating the scale of the drawing.
&scale_of_view_detailed_bar	
&sym(<symbolname>)	Includes a drawing symbol in a note, where symbolname is the name of the symbol.
&todays_date	Displays a drawing label indicating the date on which the note was created in the form dd-mm-yy (for example, 2-Jan-92). You can edit it as any other nonparametric note, using Text Line or Full Note. If you include this symbol in a format table, **Pro**|ENGINEER evaluates it when it copies the format into the drawing. To specify the initial display of the date in a drawing, use the configuration file option "todays_date_note_format."
&total_sheets	Displays a drawing label indicating the total number of sheets in the drawing.
&type	Displays a drawing label indicating the drawing model type (for example, part, assembly, etc.).
&view_name	Displays a drawing label indicating the name of the view. You *cannot* use this parameter in a drawing note. **Pro**|ENGINEER creates it with a view and places it in notes automatically. You can modify its value, but you cannot call it out in another note.
&view_scale	Displays a drawing label indicating the name of a general scaled view. You *cannot* use this parameter in a drawing note. **Pro**|ENGINEER creates it with a view and places it in notes automatically. You can modify its value, but you cannot call it out in another note.

Appendix 3: To Control the Format of the Date

The configuration file option todays_date_note_format controls the initial format of the date displayed in a drawing. The format for the setting is a string consisting of three portions: the year, the month, and the date. You can enter the portions in any order. The default value is %dd-%Mmm-%yy.

- Year
- %yy, for 97
- %yyyy, for 1997

Month (if the month contains two digits (for example, 10),% mm, % m, or % m all produce the same result)

- %Mmm, for Jan
- %MMM, for JAN
- %Month, for January
- %MONTH, for JANUARY
- %mm, for 01
- %m, for 1
- % m, for <space>1

Date (if 2 digits are needed to represent the date, all three are the same. Therefore, "%dd %mm %yy" produces "01 01 97," and "%MMM %d %yyyy" produces "JAN 1 1997")

- %dd, for 01
- %d, for 1
- % d, for <space>1.

The following formats are also valid:

- %dd-%Mmm-%yy (= 01-Jan-97)
- %mm/%dd/%yy (= 01/01/97)
- %Mmm %dd,%yyyy (= Jan 01, 1997)

Appendix 4. To create a multiple parts drawing

There will be times when an assembly drawing is not enough; you may need to create a sheet that contains detail drawings of several parts of the assembly. Of course you could create a sheet for each part but occasionally you may want to put several parts on the same drawing sheet.

1. Open the new sheet or a blank format **Right-Click** and select **Drawing Models**.

Ensure the **Layout** tab is selected in the Ribbon Interface

2. Select **Drawing models**

3. Select **Add Model**

4. The **Open** dialogue box open.

5. Open the part you want to add to the drawing.

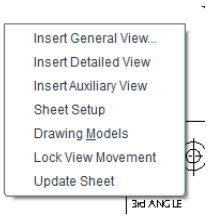
6. **Right-Click** on the sheet and select **Insert General view**.

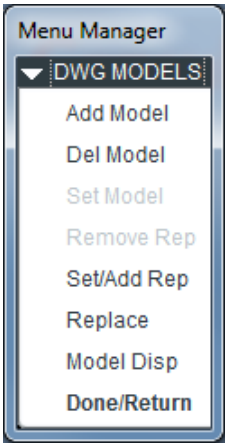
7. Proceed as for any other type of view.

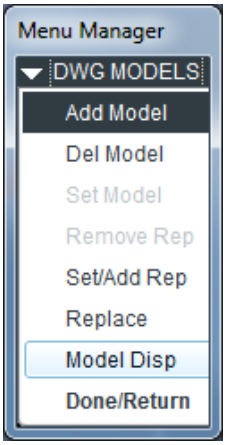
8. Repeat to add further parts to the sheet.

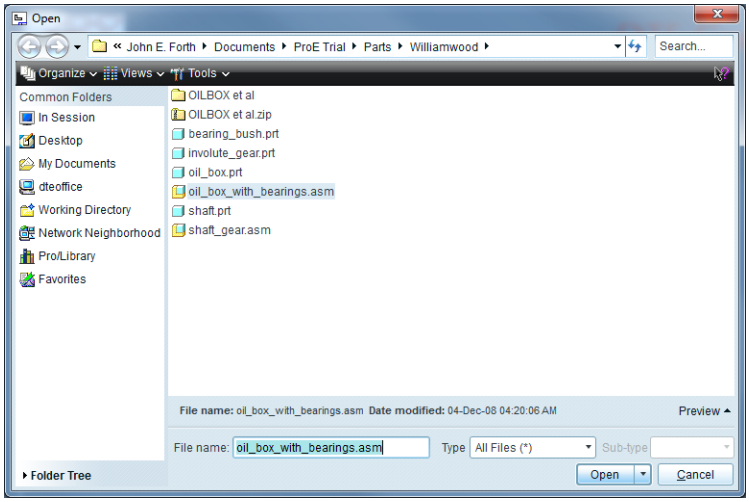
9. Note: Each part will be placed with a scale selected to fit the sheet. To ensure that the scale is known for each, check the **Custom Scale** button under the scale tab of the **Drawing View** dialogue box. This will put a scale against each part. Notes can be added as required.

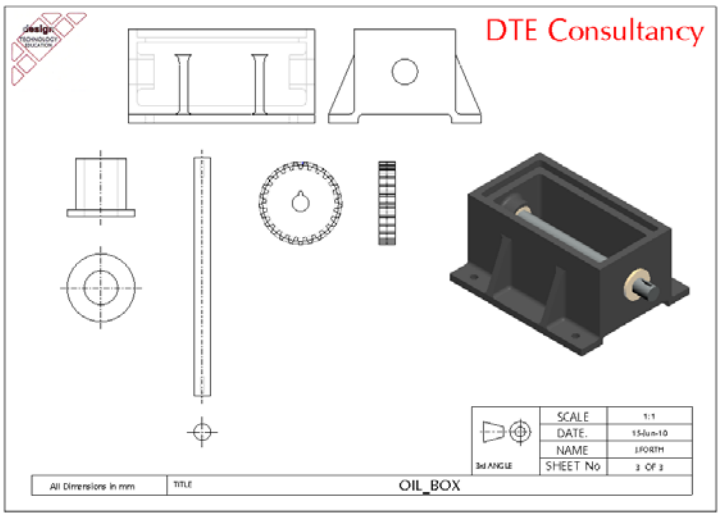
See right for an example.



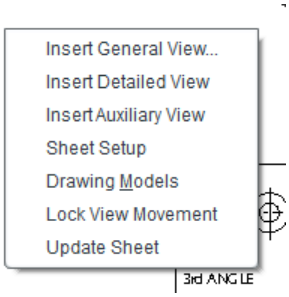




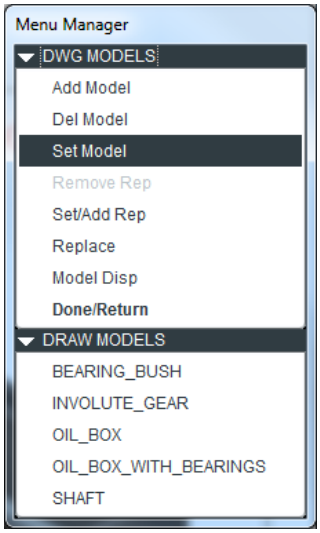




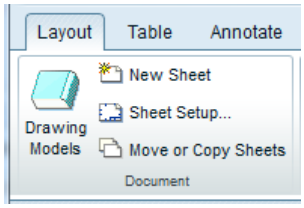
1. To set the model for use, **Right-Click** and select **Drawing Models**.



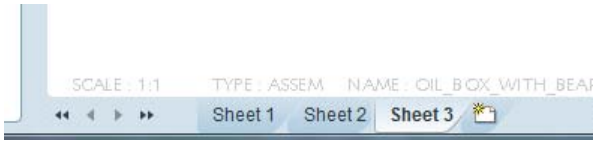
2. **Left-Click Set Model** and a list of the models associated with the drawing appear
3. **Left-Click** the model name to set.
4. **Left-Click** Done/Return



1. To Add a New Sheet, **Left-Click** New sheet on the Layout Tab.



To Move to a sheet Left-Click on the particular tab as opposite



John Forth – June 2010